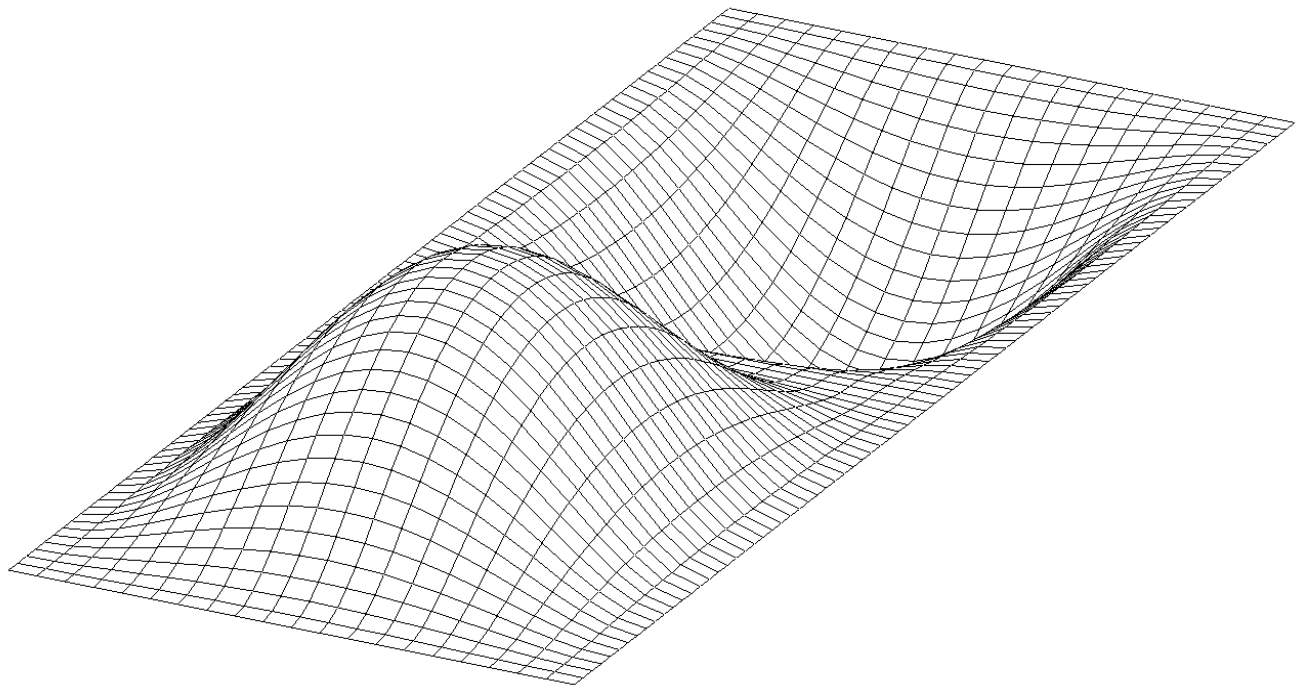


# Autodesk® Nastran® 2018

## Verification Manual



**© 2017 Autodesk, Inc. All rights reserved.**

**Autodesk® Nastran® 2018**

Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Certain materials included in this publication are reprinted with the permission of the copyright holder.

**Trademarks**

The following are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and other countries: 123D, 3ds Max, Alias, ATC, AutoCAD LT, AutoCAD, Autodesk, the Autodesk logo, Autodesk 123D, Autodesk Homestyler, Autodesk Inventor, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSketch, AutoSnap, AutoTrack, Backburner, Backdraft, Beast, BIM 360, Burn, Buzzsaw, CADmep, CAiCE, CAMduct, Civil 3D, Combustion, Communication Specification, Configurator 360, Constructware, Content Explorer, Creative Bridge, Dancing Baby (image), DesignCenter, DesignKids, DesignStudio, Discreet, DWF, DWG, DWG (design/logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DWGX, DXF, Ecotect, Ember, ESTmep, Evolver, FABmep, Face Robot, FBX, Fempro, Fire, Flame, Flare, Flint, ForceEffect, FormIt, Freewheel, Fusion 360, Glue, Green Building Studio, Heidi, Homestyler, HumanIK, i-drop, ImageModeler, Incinerator, Inferno, InfraWorks, InfraWorks 360, Instructables, Instructables (stylized robot design/logo), Inventor, Inventor HSM, Inventor LT, Lustre, Maya, Maya LT, MIMI, Mockup 360, Moldflow Plastics Advisers, Moldflow Plastics Insight, Moldflow, Moondust, MotionBuilder, Movimento, MPA (design/logo), MPA, MPI (design/logo), MPX (design/logo), MPX, Mudbox, Navisworks, ObjectARX, ObjectDBX, Opticore, Pixlr, Pixlr-o-matic, Productstream, Publisher 360, RasterDWG, RealDWG, ReCap, ReCap 360, Remote, Revit LT, Revit, RiverCAD, Robot, Scaleform, Showcase, Showcase 360, SketchBook, Smoke, Socialcam, Softimage, Sparks, SteeringWheels, Stitcher, Stone, StormNET, TinkerBox, ToolClip, Topobase, Toxik, TrustedDWG, T-Splines, ViewCube, Visual LISP, Visual, VRED, Wire, Wiretap, WiretapCentral, XSI.

NASTRAN® is a registered trademark of the National Aeronautics Space Administration. All other brand names, product names or trademarks belong to their respective holders.

**Disclaimer**

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. "AS IS." AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

## TABLE OF CONTENTS

|                                                                            |          |
|----------------------------------------------------------------------------|----------|
| <b>1. Introduction .....</b>                                               | <b>7</b> |
| 1.1 Model Files Location.....                                              | 7        |
| <b>2. Linear Statics Verification Using Theoretical Solutions.....</b>     | <b>8</b> |
| 2.1 Nodal Loads on a Cantilever Beam.....                                  | 9        |
| 2.2 Axial Distributed Load on a Linear Beam.....                           | 11       |
| 2.3 Distributed Loads on a Cantilever Beam.....                            | 13       |
| 2.4 Moment Load on a Cantilever Beam .....                                 | 15       |
| 2.5 Thermal Strain, Displacement, and Stress on a Heated Beam .....        | 17       |
| 2.6 Uniformly Distributed Load on a Linear Beam .....                      | 19       |
| 2.7 Membrane Loads on a Plate .....                                        | 21       |
| 2.8 Thin Wall Cylinder in Pure Tension.....                                | 23       |
| 2.9 Thin Shell Beam Wall in Pure Bending.....                              | 25       |
| 2.10 Strain Energy of a Truss.....                                         | 27       |
| 2.11 Flat Square Plate .....                                               | 29       |
| 2.12 The Raasch Challenge Problem for Shell Elements .....                 | 31       |
| 2.13 Twisted Beam Static Load .....                                        | 34       |
| 2.14 Plane Frame with Beam Span Loads .....                                | 36       |
| 2.15 Thermal Gradient Loads on a Beam .....                                | 38       |
| 2.16 Statically Indeterminate Reaction Force Analysis .....                | 40       |
| 2.17 Beam Stresses and Deflection .....                                    | 42       |
| 2.18 Laterally Loaded Tapered Support Structure Case 1.....                | 44       |
| 2.19 Laterally Loaded Tapered Support Structure Case 2.....                | 46       |
| 2.20 Bending of a Tee-Shaped Beam.....                                     | 48       |
| 2.21 Bending of a Circular Plate .....                                     | 50       |
| 2.22 Beam on Elastic Foundation.....                                       | 52       |
| 2.23 Thick Walled Cylinder Plain Strain.....                               | 54       |
| 2.24 Scordelis-Lo Roof .....                                               | 56       |
| 2.25 Out-of-Plane Bending of a Curved Bar .....                            | 58       |
| 2.26 Deflection of Hinged Support .....                                    | 60       |
| 2.27 Thermal Stresses in a Plate .....                                     | 62       |
| 2.28 Bending of a Tapered Plate.....                                       | 64       |
| 2.29 Bending of a Tapered Beam .....                                       | 66       |
| 2.30 Bending of a Curved Thick Beam.....                                   | 68       |
| 2.31 Truss Reaction One .....                                              | 70       |
| 2.32 Truss Reaction Two .....                                              | 72       |
| 2.33 Fixed Ended Beam Un-symmetric Tapered Member .....                    | 74       |
| 2.34 Straight Cantilever Beam Using Solids .....                           | 76       |
| 2.35 Curved Beam Using Solids .....                                        | 78       |
| 2.36 Cantilever Modeled with Variable Thickness Shells and Membranes ..... | 80       |
| 2.37 Cantilever Modeled with Variable Thickness Solids .....               | 82       |
| 2.38 Elongation of a Solid Bar .....                                       | 84       |
| 2.39 Thin Shell Beam in Pure Bending .....                                 | 86       |
| 2.40 Static Analysis of Thermal Loading.....                               | 88       |

|                                                                                       |            |
|---------------------------------------------------------------------------------------|------------|
| 2.41 Static Analysis of Pressure Loading .....                                        | 90         |
| <b>3. Linear Statics Verification Using Standard NAFEMS Benchmarks .....</b>          | <b>92</b>  |
| 3.1 Warped Element Test Cases .....                                                   | 93         |
| 3.1.1 Cylindrical Shell Patch .....                                                   | 93         |
| 3.1.2 Hemisphere-Point Loads .....                                                    | 95         |
| 3.2 Laminate Plate Element Test Cases .....                                           | 97         |
| 3.2.1 Laminated Strip .....                                                           | 97         |
| 3.2.2 Wrapped Thick Cylinder .....                                                    | 99         |
| 3.3 Shell Element Test Cases .....                                                    | 101        |
| 3.3.1 Elliptic Membrane .....                                                         | 101        |
| 3.3.2 Z-Section Cantilever .....                                                      | 103        |
| 3.3.3 Skew Plate Under Normal Pressure .....                                          | 105        |
| <b>4. Normal Modes/Eigenvalue Verification Using Theoretical Solutions .....</b>      | <b>107</b> |
| 4.1 Two Degree of Freedom Undamped Free Vibration – Principle Modes .....             | 108        |
| 4.2 Three Degree of Freedom Torsional System .....                                    | 110        |
| 4.3 Cantilever Plate Eigenvalue Problem .....                                         | 112        |
| 4.4 Bathe and Wilson Frame Eigenvalue Problem .....                                   | 114        |
| 4.5 Natural Frequency of a Cantilevered Mass .....                                    | 116        |
| 4.6 Fundamental Frequency of a Simply Supported Beam .....                            | 118        |
| 4.7 Natural Frequencies of a Cantilever Beam .....                                    | 120        |
| 4.8 Vibration of a String Under Tension .....                                         | 122        |
| 4.9 Vibration of a Wedge .....                                                        | 124        |
| <b>5. Normal Modes/Eigenvalue Verification Using Standard NAFEMS Benchmarks .....</b> | <b>126</b> |
| 5.1 Bar Element Test Cases .....                                                      | 127        |
| 5.1.1 Pin-ended Cross – In-plane Vibration .....                                      | 127        |
| 5.1.2 Free Square Frame – In-plane Vibration .....                                    | 129        |
| 5.1.3 Cantilever with Off-center Point Masses .....                                   | 131        |
| 5.1.4 Deep Simply-Supported Beam .....                                                | 133        |
| 5.1.5 Pin-ended Double Cross – In-plane Vibration .....                               | 135        |
| 5.1.6 Circular Ring – In-plane and Out-of-plane Vibration .....                       | 137        |
| 5.2 Plate Element Test Cases .....                                                    | 139        |
| 5.2.1 Thin Square Cantilevered Plate – Symmetric Modes .....                          | 139        |
| 5.2.2 Thin Square Cantilevered Plate – Anti-symmetric Modes .....                     | 142        |
| 5.2.3 Free Thin Square Plate .....                                                    | 144        |
| 5.2.4 Simply – Supported Thin Square Plate .....                                      | 146        |
| 5.2.5 Simply – Supported Thin Annular Plate .....                                     | 148        |
| 5.2.6 Clamped Thin Rhombic Plate .....                                                | 150        |
| 5.2.7 Simply – Supported Thick Square Plate, Test A .....                             | 152        |
| 5.2.8 Simply – Supported Thick Square Plate, Test B .....                             | 154        |
| 5.2.9 Clamped Thick Rhombic Plate .....                                               | 156        |
| 5.2.10 Simply – Supported Thick Annular Plate .....                                   | 158        |
| 5.2.11 Cantilevered Square Membrane .....                                             | 160        |
| 5.2.12 Cantilevered Tapered Membrane .....                                            | 162        |
| 5.2.13 Free Annular Membrane .....                                                    | 165        |
| 5.3 Axisymmetric Solid and Solid Element Test Cases .....                             | 167        |
| 5.3.1 Simply – Supported “Solid” Square Plate .....                                   | 167        |
| <b>6. Verification Test Cases from the Societe Francaise des Mechaniciens .....</b>   | <b>170</b> |
| 6.1 Mechanical Structures – Linear Statics Analysis with Bar or Rod Elements .....    | 171        |
| 6.1.1 Short Beam on Two Articulated Supports .....                                    | 171        |
| 6.1.2 Clamped Beams Linked by a Rigid Element .....                                   | 173        |
| 6.1.3 Plane Bending Load on a Thin Arc .....                                          | 175        |
| 6.1.4 Articulated Plane Truss .....                                                   | 177        |

|                                                                                           |            |
|-------------------------------------------------------------------------------------------|------------|
| <b>6.2 Mechanical Structures – Linear Statics Analysis with Plate Elements.....</b>       | <b>179</b> |
| 6.2.1 Plane Shear and Bending Load on a Plate .....                                       | 179        |
| 6.2.2 Uniformly Distributed Load on a Circular Plate .....                                | 181        |
| 6.2.3 Torque Loading on a Square Tube .....                                               | 183        |
| 6.2.4 Cylindrical Shell with Internal Pressure .....                                      | 185        |
| 6.2.5 Gravity Loading on a Thin Wall Cylinder .....                                       | 187        |
| 6.2.6 Pinched Cylindrical Shell .....                                                     | 189        |
| 6.2.7 Infinite Plate with a Circular Hole .....                                           | 191        |
| <b>6.3 Mechanical Structures – Linear Statics Analysis with Solid Elements .....</b>      | <b>193</b> |
| 6.3.1 Thick Plate Clamped at Edges .....                                                  | 193        |
| <b>6.4 Mechanical Structures – Normal Modes/Eigenvalue Analysis .....</b>                 | <b>196</b> |
| 6.4.1 Cantilever Beam with a Variable Rectangular Section .....                           | 196        |
| 6.4.2 Thin Circular Ring Clamped at Two Points .....                                      | 198        |
| 6.4.3 Vibration Modes of a Thin Pipe Elbow .....                                          | 200        |
| 6.4.4 Thin Square Plate (Clamped or Free) .....                                           | 203        |
| 6.4.5 Simply – Supported Rectangular Plate .....                                          | 206        |
| 6.4.6 Bending of a Symmetric Truss .....                                                  | 208        |
| 6.4.7 Hovgaard’s Problem – Pipes with Flexible Elbows .....                               | 210        |
| 6.4.8 Rectangular Plates .....                                                            | 212        |
| <b>6.5 Stationary Thermal Tests – Steady-State Heat Transfer Analysis .....</b>           | <b>214</b> |
| 6.5.1 L-Plate.....                                                                        | 214        |
| <b>7. Buckling Verification Using Theoretical Solutions.....</b>                          | <b>216</b> |
| 7.1 Buckling of a Thin Walled Cylinder .....                                              | 217        |
| 7.2 Buckling of a Bar with Hinged Ends .....                                              | 219        |
| 7.3 Buckling of a Bar with Hinged Ends Using Plates .....                                 | 221        |
| 7.4 Buckling of a Bar with Hinged Ends Using Solids .....                                 | 223        |
| 7.5 Buckling of a Rectangular Plate Under Concentrated Center Loads.....                  | 225        |
| 7.6 Buckling of a Rectangular Plate Under End Uniform Load .....                          | 227        |
| <b>8. Dynamics Verification Using Standard NAFEMS Benchmarks.....</b>                     | <b>229</b> |
| 8.1 Transient Forced Vibration Response – Deep Simply Supported Beam.....                 | 230        |
| 8.2 Periodic Forced Vibration Response – Deep Simply Supported Beam.....                  | 232        |
| 8.3 Modal Transient Forced Vibration Response – Simply Supported Plate .....              | 235        |
| 8.4 Harmonic Forced Vibration Response – Simply Supported Plate .....                     | 237        |
| 8.5 Random Forced Vibration Response – Simply Supported Plate.....                        | 239        |
| 8.6 Direct Transient Forced Vibration Response – Simply Supported Plate.....              | 241        |
| 8.7 Direct Frequency Response – Simply Supported Plate .....                              | 244        |
| <b>9. Dynamics Verification Using Theoretical Solutions .....</b>                         | <b>246</b> |
| 9.1 Seismic Response of a Beam Structure.....                                             | 247        |
| 9.2 Cantilever Beam Subjected to Sine Plus Tip Load.....                                  | 249        |
| 9.3 Simply Supported Beam, Ramped Nodal Forcing Function, Transient Forced Vibration..... | 252        |
| 9.4 Tower Structure Under a Harmonic Excitation Force .....                               | 254        |
| 9.5 Simply Supported Beam Subjected to a Traveling Dynamic Load .....                     | 257        |
| <b>10. Nonlinear Static Verification Using Theoretical Solutions .....</b>                | <b>260</b> |
| 10.1 Nonlinear Cable Tension .....                                                        | 261        |
| 10.2 Cable Supporting Hanging Loads .....                                                 | 263        |
| 10.3 Ten Story Plane Frame .....                                                          | 265        |
| 10.4 Straight Cantilever with Axial End Point Load .....                                  | 267        |
| 10.5 Residual Stress Problem.....                                                         | 269        |
| <b>11. Nonlinear Static Verification Using Standard NAFEMS Benchmarks .....</b>           | <b>271</b> |

- 11.1 Elastic Large Deformation Response of a Z-shaped Cantilever Under an End Load ..... 272
- 11.2 Straight Cantilever with End Moment ..... 274
- 11.3 Lee’s Frame Buckling Problem ..... 276
- 12. Nonlinear Dynamic Verification Using Theoretical Solutions..... 278
  - 12.1 Impact Load on a Rod by a Mass at a Constant Velocity ..... 279
  - 12.2 Impact of a Block on a Spring Scale..... 282
- APPENDIX A – REFERENCES ..... 284

## 1. Introduction

This guide contains verification test cases for the Autodesk® Nastran® Finite Element Analysis solver. These test cases verify the functionality of Autodesk® Nastran® and encompass the different analysis types using theoretical and benchmark solutions from well-known engineering test cases. Each test case contains all test data needed to reproduce the given results. This guide contains test cases for:

- Linear Statics verification using theoretical solutions
- Normal Modes/Eigenvalue verification using theoretical solutions
- Normal Modes/Eigenvalue verification using standard NAFEMS benchmarks
- Verification Test Cases from the Societe Francaise des Mechaniciens

### 1.1 Model Files Location

All the Autodesk Nastran model files (.nas) used for these test cases are located in this folder:

C:\Users\Public\Documents\Autodesk\Nastran 2018\Example Files\en-us\Verification Models

## 2. Linear Statics Verification Using Theoretical Solutions

The purpose of these linear static test cases is to verify the functionality of Autodesk Nastran using theoretical solutions of well-known engineering linear static problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

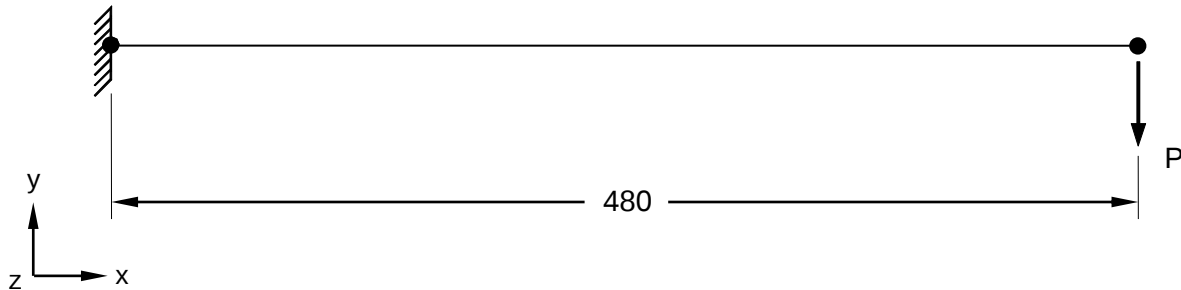
For most cases, discrepancies between Autodesk Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.



## 2.1 Nodal Loads on a Cantilever Beam

### Problem Description

Figure 1 shows the cantilever beam with a load acting on the free end. A static analysis is performed on the model. Beam deflection at the free end of the beam, and shear stress at the constrained end of the beam are determined. All dimensions are in inches.



**Figure 1. Cantilever Beam with Nodal Load**

### Autodesk Nastran Analysis Model Filename

- vm2\_1.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (4 x 1): 5 nodes, 4 bar elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 480$  in

#### *Cross Sectional Properties*

Area:  $A = 900$  in<sup>2</sup>

Square Cross Section = (30 in x 30 in)

Moment of Inertia:  $I_y = I_z = 67500$  in<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+6 psi

#### *Boundary Conditions*

One end of the beam is constrained in all translations and rotations. A load  $P = 50,000$  lb force in the negative Y-direction is set at the free end of the beam.

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                     | Theory  | Autodesk Nastran | Error (%) |
|-------------------------------------------------|---------|------------------|-----------|
| Beam Constrained End Z Shear Force Stress (psi) | 5333.3  | 5333.3           | 0.0       |
| Beam Free End T2 Translation (in)               | 0.9130* | 0.9130           | 0.0       |

**\*Note:** The original theoretical value of 0.9102 neglects the shear deformation. The value for shear deformation is calculated below and added to the original theoretical value.

**Post Processing**

$$\text{Shear Deformation} = \frac{VL}{AG}$$

where: V = Shear Load, L = Length of the beam, A = Shear area = 0.8333 x cross section area, and G = modulus of rigidity

$$\text{Shear deformation} = 0.0028$$

Adding the shear deformation to the theory, T2 Translation = (Theory value + shear deformation)

$$\text{T2 Translation} = (0.9102 + 0.0028) = 0.9130 \text{ in}$$

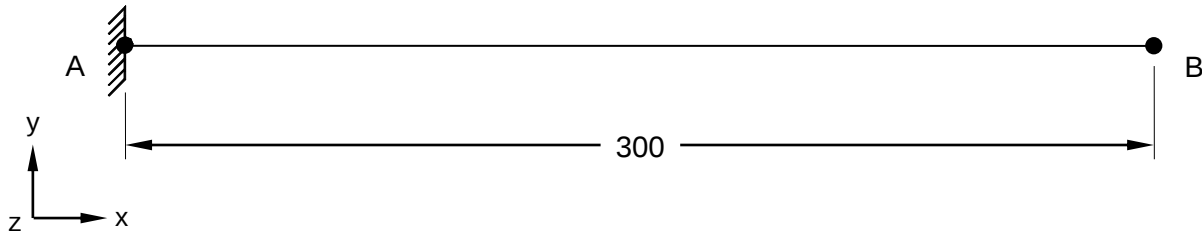
**References**

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992. p.716.

## 2.2 Axial Distributed Load on a Linear Beam

### Problem Description

Figure 1 shows the model of the linear beam. A static analysis is performed using an axially distributed load. The beam axial stress at the constrained end (A), deflection at the free end (B), and the constraint force at the constrained end of the beam are determined. All dimensions are in inches.



**Figure 1. Linear Beam with Axial Distributed Load**

### Autodesk Nastran Analysis Model Filename

- vm2\_2.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (30 x 1): 31 nodes, 30 bar elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 300$  in

#### *Cross Sectional Properties*

Area:  $A = 9$  in<sup>2</sup>

Square Cross Section = (3 in x 3 in)

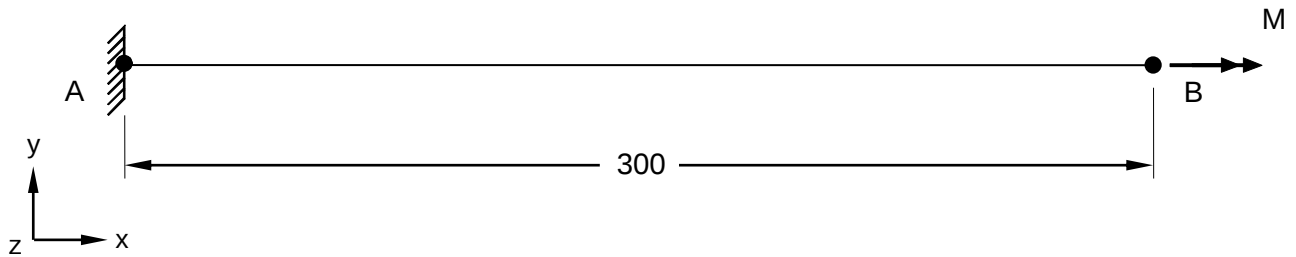
Moment of Inertia:  $I = 6.75$  in<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+6 psi

### Boundary Conditions

One end of the beam is constrained in all translations and rotations (point A). An axially distributed load (force per unit length) is set to 1,000 lb/in in the negative Y-direction for the 10-inch long element furthest from the constrained end (at point B). See Figure 2.



**Figure 2. Boundary Conditions**

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                             | Theory   | Autodesk Nastran | Error (%) |
|-----------------------------------------|----------|------------------|-----------|
| Beam Constrained End Axial Stress (psi) | 1,111.1  | 1,111.1          | 0.0       |
| Beam Free End T1 Translation (in)       | 0.011111 | 0.010926         | 1.7       |
| Beam T1 Constraint Force (lb)           | -10,000  | -10,000          | 0.0       |

### References

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.

## 2.3 Distributed Loads on a Cantilever Beam

### Problem Description

Figure 1 shows the model of a cantilever beam with a distributed load acting in the negative Y-direction. A static analysis is performed on the model. The beam torque stress at the constrained end of the beam (A), the deflection of the free end (B), and the total beam constraint force are determined. All dimensions are in inches.

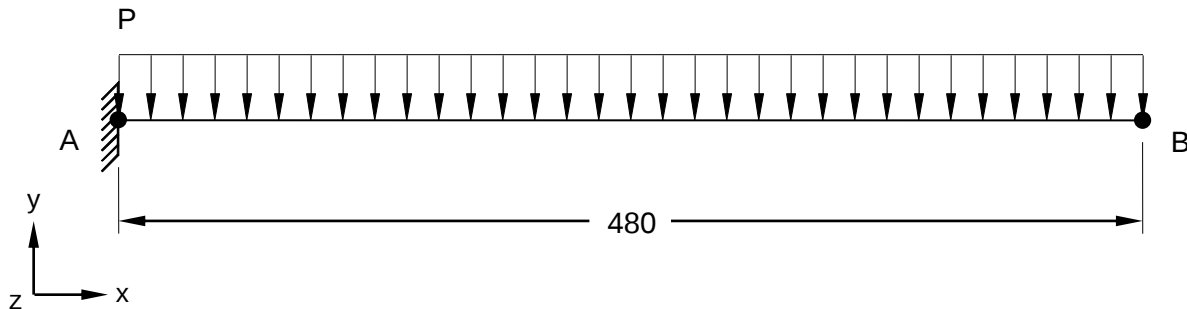


Figure 1. Cantilever Beam with Distributed Load

### Autodesk Nastran Analysis Model Filename

- vm2\_3.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (8 x 1): 9 nodes, 8 bar elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 480$  in

#### *Cross Sectional Properties*

Area:  $A = 900$  in<sup>2</sup>

Square Cross Section = (30 in x 30 in)

Moment of Inertia:  $I_y = I_z = 67500$  in<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+6 psi

**Boundary Conditions**

One end of the beam is constrained in all translations and rotations. A distributed load  $P = 250$  lb/in in the negative Y-direction is defined on all elements.

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                              | Theory  | Autodesk Nastran | Error (%) |
|------------------------------------------|---------|------------------|-----------|
| Beam Constrained End Torque Stress (psi) | 6,400   | 6,400            | 0.00      |
| Beam Free End Total Translation (in)     | 0.8192* | 0.8192           | 0.00      |
| Beam Total Constraint Force (lb)         | 120,000 | 120,000          | 0.00      |

**\*Note:** The theoretical value neglects the shear deformation.

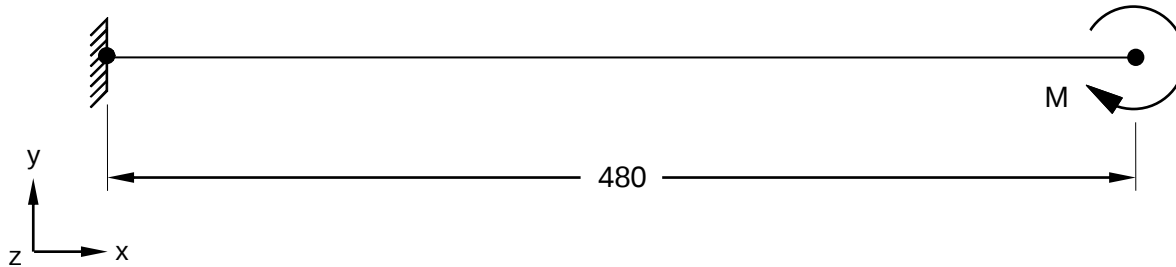
**References**

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.

## 2.4 Moment Load on a Cantilever Beam

### Problem Description

Figure 1 shows the model of a cantilever beam with moment load acting on the free end of the beam. A static analysis is performed on the model. The beam bending stress at the constrained end of the beam, the deflection at the free end, and the reaction force at the constrained end are determined. All dimensions are in inches.



**Figure 1. Cantilever Beam with Moment Load**

### Autodesk Nastran Analysis Model Filename

- vm2\_4.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (8 x 1): 9 nodes, 8 bar elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 480$  in

#### *Cross Sectional Properties*

Area:  $A = 900$  in<sup>2</sup>

Square Cross Section = (30 in x 30 in)

Moment of Inertia:  $I_y = I_z = 67500$  in<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+6 psi

#### *Boundary Conditions*

One end of the beam is constrained in all translations and rotations. A  $M = 2.5$  E+6 in-lb is set at the free end of the beam in the Z-direction.

***Solution Type***

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                       | Theory    | Autodesk Nastran | Error (%) |
|---------------------------------------------------|-----------|------------------|-----------|
| Beam Constrained End Z Shear Force Stress (psi)   | 555.6     | 555.6            | 0.0       |
| Beam Free End Total Translation (in)              | 0.1422    | 0.1422           | 0.0       |
| Beam Constrained End Total Constraint Moment (lb) | 2,500,000 | 2,500,000        | 0.0       |

**References**

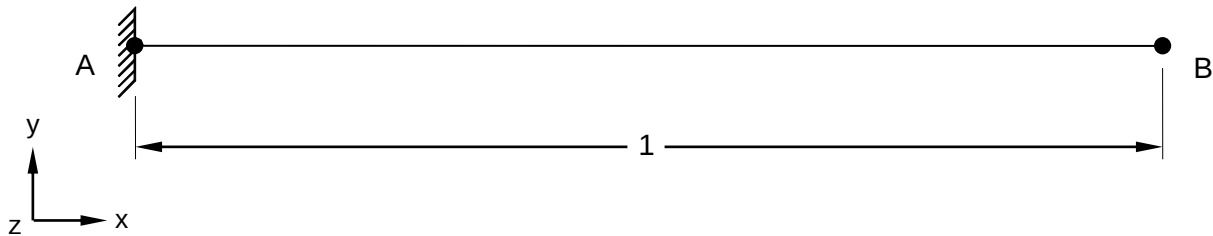
1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.



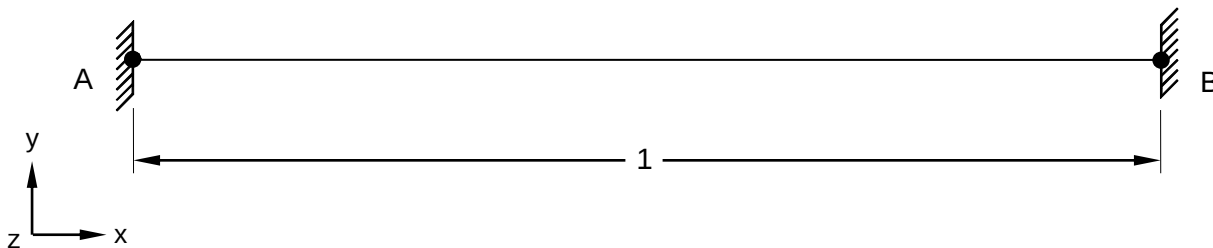
## 2.5 Thermal Strain, Displacement, and Stress on a Heated Beam

### Problem Description

Figure 1 shows the model of a heated beam. A static analysis is performed on the model. In one case of this example, the beam is constrained at one end. For the second case, the beam is constrained at both ends (see Figure 2). The beam axial stress at the constrained end of the beam, the deflection at the free end, and the constraint force at the constrained end of the beam are determined. All dimensions are in meters.



**Figure 1. Heated Beam with Thermal Strain, Displacement, and Stress (Case 1)**



**Figure 2. Heated Beam with Thermal Strain, Displacement, and Stress (Case 2)**

### Autodesk Nastran Analysis Model Filename

- vm2\_5a.nas – Case 1 (beam constrained at one end)
- vm2\_5b.nas – Case 2 (beam constrained at both ends)

### Model Data

#### *Finite Element Modeling*

- Mesh (10 x 1): 11 nodes, 10 bar elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 1$  m

### Cross Sectional Properties

Area:  $A = 0.01 \text{ m}^2$

### Material Properties

Young's Modulus:  $E = 2.068 \text{ E}+11 \text{ Pa}$

Thermal Expansion Coefficient:  $\alpha = 1.2 \text{ E}-05 \text{ m}/(\text{m } ^\circ\text{C})$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Case 1: One end of the beam is constrained in all translations and rotations. The temperature on all nodes is set to 25°C. The reference temperature is set to -50°C.

Case 2: Both ends of the beam are constrained in all translations and rotations. The temperature on all nodes is set to 25°C. The reference temperature is set to -50°C.

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1 and Table 2.

**Table 1. Results for Case 1**

| Description                         | Theory | Autodesk Nastran | Error (%) |
|-------------------------------------|--------|------------------|-----------|
| Beam Free End Total Translation (m) | 0.0009 | 0.0009           | 0.0       |
| Beam Constrained End Axial Strain   | 0.0009 | 0.0009           | 0.0       |

**Table 2. Results for Case 2**

| Description                                      | Theory | Autodesk Nastran | Error (%) |
|--------------------------------------------------|--------|------------------|-----------|
| Beam Total Translation (m)                       | 0.0    | 0.0              | 0.0       |
| Beam Constrained End Total Constraint Force (kN) | 1,860  | 1,860            | 0.0       |
| Beam Constrained End Axial Stress (MPa)          | -186   | -186             | 0.0       |

### References

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.

## 2.6 Uniformly Distributed Load on a Linear Beam

### Problem Description

Figure 1 shows the model of a beam with distributed loads acting on both ends and constraints acting on the second node in from either end (B and D). A static analysis is performed on the model. The beam torque stress at the center of the beam and the deflection at the center of the beam are determined. All dimensions are in inches.

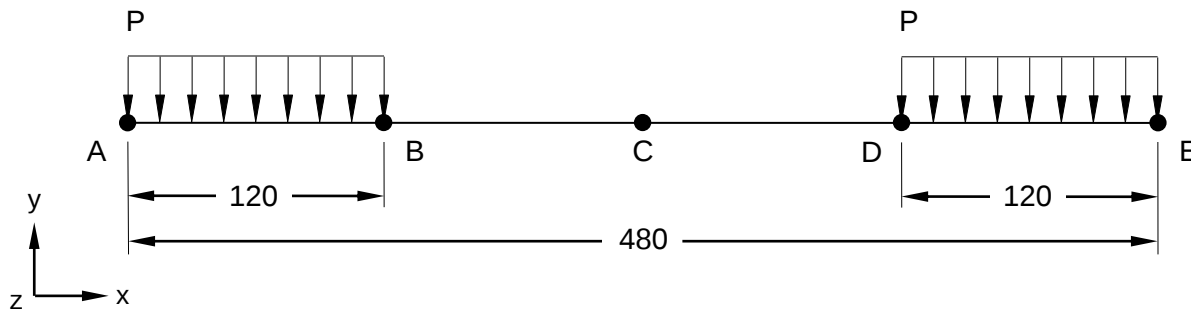


Figure 1. Linear Beam with Uniformly Distributed Load

### Autodesk Nastran Analysis Model Filename

- vm2\_6.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (4 x 1): 5 nodes, 4 bar elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 480$  in

#### *Cross Sectional Properties*

Area:  $A = 50.5908$  in<sup>2</sup>

Rectangular Cross Section = (1.17 in x 43.24 in)

Moment of Inertia:  $I_z = 7892$  in<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+6 psi

**Boundary Conditions**

The second node in from either end of the beam (B and D, as shown in Figure 1) is constrained in all translations and X and Y-rotations. Rotation about Z is not constrained. A distributed load  $P = 833 \text{ lb/in}$  is defined in the negative Y-direction on the end elements of the beam.

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                        | Theory | Autodesk Nastran | Error (%) |
|------------------------------------|--------|------------------|-----------|
| Beam Center Total Translation (in) | 0.182  | 0.182            | 0.0       |
| Beam Center Torque Stress (psi)    | 16,439 | 16,439           | 0.0       |

**References**

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955. p.98.

## 2.7 Membrane Loads on a Plate

### Problem Description

Figure 1 shows the plate with membrane loads acting in the plane of the plate. Static analysis is performed on the model. The change in length of diameter AB and of diameter CD are determined. All dimensions are in inches.

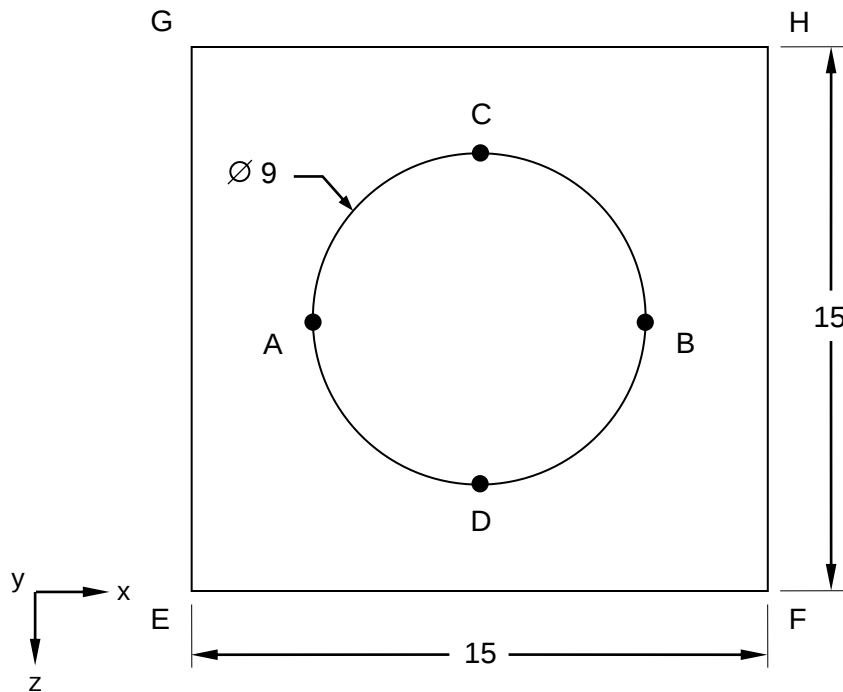


Figure1. Plate with Membrane Loads

### Autodesk Nastran Analysis Model Filename

- vm2\_7.nas

### Model Data

#### *Finite Element Modeling*

- A  $\frac{1}{4}$  of the model is created (the bottom right-hand corner that contains points B and D). Symmetry boundary conditions are applied. The answer is then multiplied by 2 for correct results.

### Units

inch/pound/second

**Model Geometry**

Length:  $L = 15$  in

Diameter:  $AB, CD = 9$  in

Thickness:  $t = 3/4$  in

**Material Properties**

Young's Modulus:  $E = 10.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.3$

$F(x)/I = 9,000$  lb/in

$F(z)/I = 15,000$  lb/in

**Boundary Conditions**

The model is constrained using symmetry boundary conditions (on edges EG and GH, as shown in Figure 1). The elemental edge load is set to 9,000 lb/in in the X-direction (on edge FH) and 15,000 lb/in in the negative Y-direction (on edge EF).

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description         | Theory | Autodek Nastran | Error (%) |
|---------------------|--------|-----------------|-----------|
| T1 Translation (in) | 0.0048 | 0.0048          | 0.0       |
| T3 Translation (in) | 0.0144 | 0.0144          | 0.0       |

**References**

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992. p.85.

## 2.8 Thin Wall Cylinder in Pure Tension

### Problem Description

Figure 1 shows the thin wall cylinder with uniform axial loads. Static analysis is performed on the model. The stress and deflection of the thin wall cylinder is determined. All dimensions are in inches. The wall thickness is 0.01 inches.

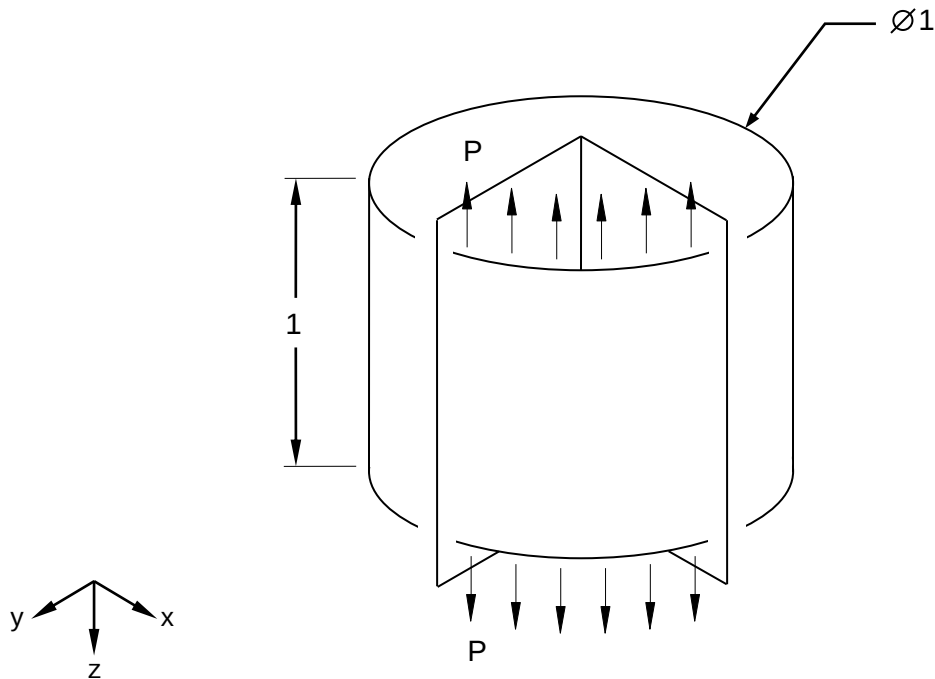


Figure 1. Thin Wall Cylinder in Pure Tension

### Autodesk Nastran Analysis Model Filename

- vm2\_8.nas

### Model Data

#### *Finite Element Modeling*

- 25 nodes. A  $\frac{1}{4}$  model is created with 16 5-DOF/node quadrilateral plate elements and symmetry boundary conditions.

#### *Units*

inch/pound/second

**Model Geometry**

Radius:  $R = 0.5$  in

Thickness:  $t = 0.01$  in

Height:  $h = 1.0$  in

**Material Properties**

Young's Modulus:  $E = 10.0 \text{ E}+3$  psi

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

The quarter model is constrained using symmetry boundary conditions. The nodes on the edge opposite to the loaded edge are constrained in the Z-translation. The nodal force  $P/(\pi \cdot D) = 3.1831$ , where  $P = 10$  psi, is applied to nodes 21, 25 as 0.9757 lbs and nodes 22, 23, 24 as 1.9509 lbs.

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description               | Theory  | Autodesk Nastran | Error (%) |
|---------------------------|---------|------------------|-----------|
| Top Y Normal Stress (psi) | 1,000.0 | 1,000.0          | 0.0       |
| T3 Translation (in)       | 0.1     | 0.1              | 0.0       |
| T1 Translation (in)       | -0.015  | -0.015           | 0.0       |

**References**

1. Roark, R. and Young, W., *Formulas for Stress and Strain*, 6<sup>th</sup> Edition. New York: McGraw-Hill Book Co., 1989. p. 518, Case 1a.



## 2.9 Thin Shell Beam Wall in Pure Bending

### Problem Description

Figure 1 shows the thin shell beam wall. A distributed load is acting on the free end (edge CD). Static analysis is performed on the model. Maximum stress at the constrained end, maximum deflection at the free end, and the beams strain energy are determined. All dimensions are in inches.

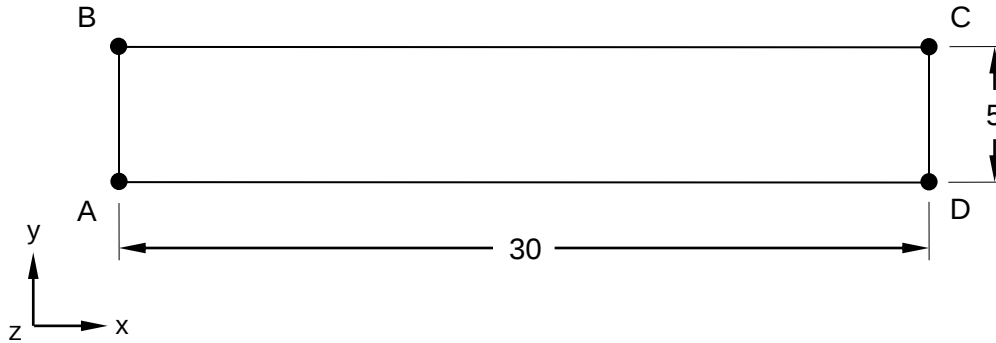


Figure 1. Thin Shell Beam Wall in Pure Bending

### Autodesk Nastran Analysis Model Filename

- vm2\_9.nas

### Model Data

#### Finite Element Modeling

- 14 nodes, 6 5-DOF/node quadrilateral plate elements

### Units

inch/pound/second

### Model Geometry

Length:  $L = 30$  in

Width:  $w = 5$  in

Thickness:  $t = 0.1$  in

### Material Properties

Young's Modulus:  $E = 30.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The nodes at end AB (nodes 7 and 14) are constrained in all translations and rotations. Distributed load as nodal forces of  $p/w = 1.2$  lbs/in, where  $p = 6.0$  lb, are applied to the opposite end of the beam, at end CD (nodes 1 and 8), acting in the negative Z-direction.

***Solution Type***

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                     | Theory | Autodesk Nastran | Error (%) |
|---------------------------------|--------|------------------|-----------|
| T3 Translation (in)             | 4.320  | 4.255            | 1.5       |
| Plate Bottom Major Stress (psi) | 21,600 | 20,535           | 4.9       |
| Total Strain Energy (lb-in)     | 12.96  | 12.77            | 1.5       |

**References**

1. Shigley, J. and Mitchel L., *Mechanical Engineering Design*, 4<sup>th</sup> Edition. New York: McGraw-Hill, Inc., 1983. pp. 134, 804.

## 2.10 Strain Energy of a Truss

### Problem Description

Figure 1 shows the truss with a load acting on the center, at point D. Static analysis is performed on the truss. The strain energy of the truss is determined. All dimensions are in inches.

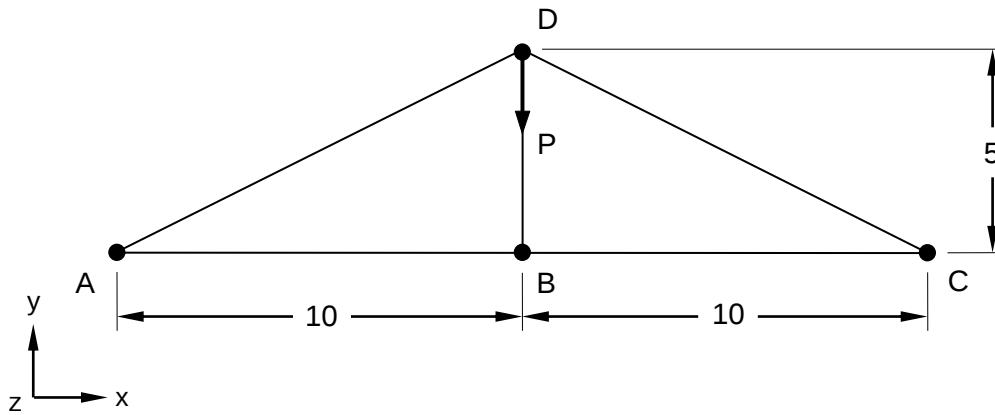


Figure 1. Truss Model

### Autodesk Nastran Analysis Model Filename

- vm2\_10.nas

### Model Data

#### *Finite Element Modeling*

- 4 nodes, 5 rod elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 10$  in

#### *Cross Sectional Properties*

Area:  $A = 0.01$  in<sup>2</sup>

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+6 psi

**Boundary Conditions**

The bottom left corner (end A) of the truss is constrained in the X, Y, and Z-translations and the X and Y rotations. The bottom right corner (end C) of the truss is constrained in the Y and Z-translations and the X and Y-rotations. The top of the truss (point D) is loaded with a nodal force  $P = 300$  lb in the negative Y-direction.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                       | Theory | Autodesk Nastran | Error (%) |
|-----------------------------------|--------|------------------|-----------|
| Truss Total Strain Energy (lb-in) | 5.846  | 5.846            | 0.0       |

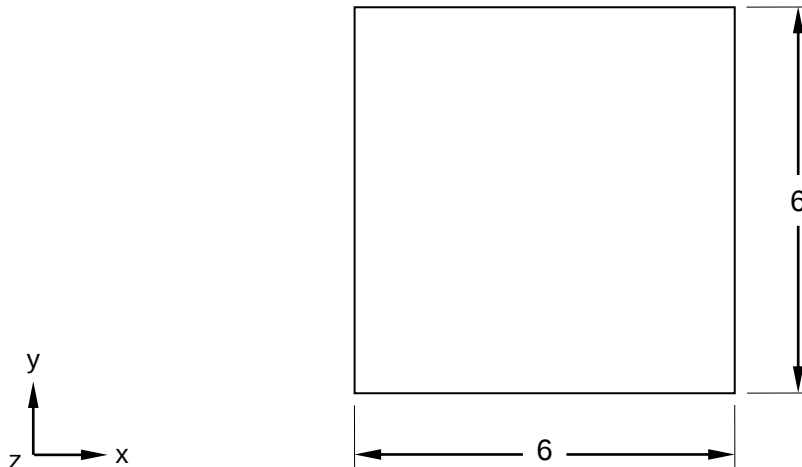
**References**

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw –Hill, Inc., 1992. p. 588.

## 2.11 Flat Square Plate

### Problem Description

Figure 1 shows the flat square plate. Static analysis is performed on the plate using two different plate thicknesses. The maximum displacement is determined. All dimensions are in inches.



**Figure 1. Flat Square Plate Model**

### Autodesk Nastran Analysis Model Filename

- vm2\_11a.nas – thick plate
- vm2\_11b.nas – thin plate

### Model Data

#### *Finite Element Modeling*

- A  $\frac{1}{4}$  model is created with 16 5-DOF/node quadrilateral plate elements and symmetry boundary conditions where applicable.

#### *Units*

inch/pound/second

#### *Model Geometry*

*Length:*  $L = 6$  in

*Width:*  $w = 6$  in

*Thickness:*  $t = 0.25$  in (vm2\_11a) and  $t = 0.0025$  in (vm2\_11b)

#### *Material Properties*

*Young's Modulus:*  $E = 29.0 \text{ E}+6$  psi

### Boundary Conditions

The quarter model is constrained using symmetry boundary conditions. A pressure load of 100 psi is applied on the thick plate (vm2\_11a) and a pressure load of 0.0001 psi is applied on the thin plate (vm2\_11b).

### Solution Type

Static

### Comparison of Results

Tabular results are given in Table 1 for the thick plate example and in Table 2 for the thin plate example.

**Table 1. Results Thick Plate**

| Description                          | Theory  | Autodesk Nastran | Error (%) |
|--------------------------------------|---------|------------------|-----------|
| Max Displacement in Z-direction (in) | 3.94E-3 | 4.01E-3          | 1.8       |

**Table 2. Results Thin Plate**

| Description                          | Theory  | Autodesk Nastran | Error (%) |
|--------------------------------------|---------|------------------|-----------|
| Max Displacement in Z-direction (in) | 3.94E-3 | 3.78E-3          | 4.1       |

### References

1. Roark, R. and Young, W., *Formulas for Stress and Strain*, 6<sup>th</sup> Edition. New York: McGraw-Hill Book Co., 1989. p. 508.

## 2.12 The Raasch Challenge Problem for Shell Elements

### Problem Description

Figure 1 shows the clamped Raasch Hook with a tip in-plane shear load. The structure consists of two cylindrical shells with different curvatures. Static analysis is performed on the plate using five different mesh sizes to check the results convergence and to be consistent with the meshes presented by Knight (1997). The maximum displacement is determined at the tip in the direction of the load. All dimensions are in inches.

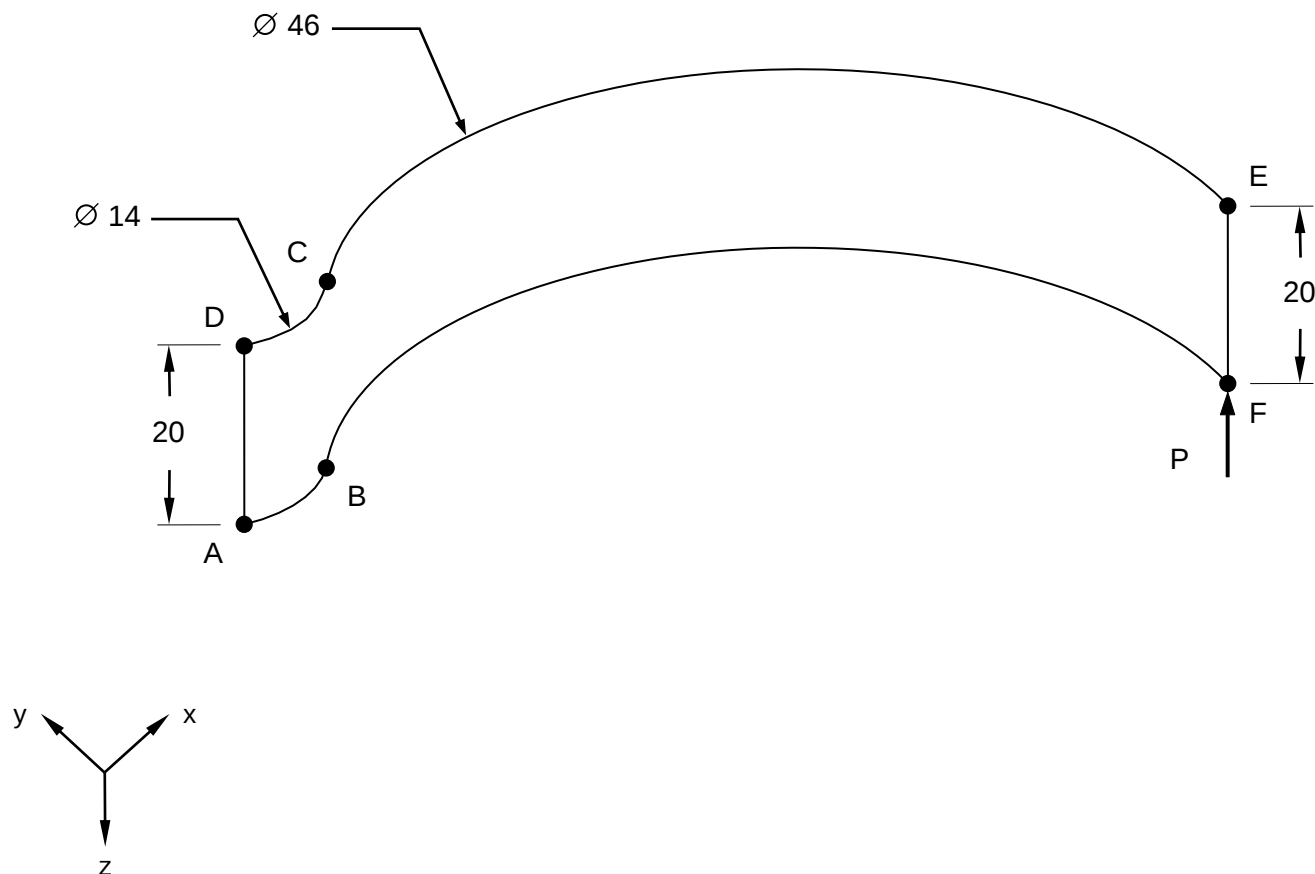


Figure 1. Raasch Hook Plate Model

### Autodesk Nastran Analysis Model Filename

- vm2\_12a4.nas – 5-DOF/node, CQUAD4 elements
- vm2\_12aR.nas – 6-DOF/node, CQUADR elements
- vm2\_12b4.nas – 5-DOF/node, CQUAD4 elements
- vm2\_12bR.nas – 6-DOF/node, CQUADR elements
- vm2\_12c4.nas – 5-DOF/node, CQUAD4 elements
- vm2\_12cR.nas – 6-DOF/node, CQUADR elements
- vm2\_12d4.nas – 5-DOF/node, CQUAD4 elements
- vm2\_12dR.nas – 6-DOF/node, CQUADR elements
- vm2\_12e4.nas – 5-DOF/node, CQUAD4 elements
- vm2\_12eR.nas – 6-DOF/node, CQUADR elements

## Model Data

### *Finite Element Modeling*

- Test vm2\_12a4: 1x9 mesh, 20 nodes, 9 5-DOF/node quadrilateral plate elements
- Test vm2\_12aR: 1x9 mesh, 20 nodes, 9 6-DOF/node quadrilateral plate elements
- Test vm2\_12b4: 3x17 mesh, 90 nodes, 51 5-DOF/node quadrilateral plate elements
- Test vm2\_12bR: 3x17 mesh, 90 nodes, 51 6-DOF/node quadrilateral plate elements
- Test vm2\_12c4: 5x34 mesh, 248 nodes, 170 5-DOF/node quadrilateral plate elements
- Test vm2\_12cR: 5x34 mesh, 248 nodes, 170 6-DOF/node quadrilateral plate elements
- Test vm2\_12d4: 10x68 mesh, 828 nodes, 680 5-DOF/node quadrilateral plate elements
- Test vm2\_12dR: 10x68 mesh, 828 nodes, 680 6-DOF/node quadrilateral plate elements
- Test vm2\_12e4: 20x136 mesh, 3014 nodes, 2720 5-DOF/node quadrilateral plate elements
- Test vm2\_12eR: 20x136 mesh, 3014 nodes, 2720 6-DOF/node quadrilateral plate elements

### *Units*

inch/pound/second

### *Model Geometry*

Radius:  $R_1 = 14$  in and  $R_2 = 46$  in

Width:  $w = 20$  in

Thickness:  $t = 2$  in

### *Material Properties*

Young's Modulus:  $E = 3300.0$  psi

Poisson's Ratio:  $\nu = 0.35$

### *Boundary Conditions*

One end is fixed in all translations and rotations (edge AB from Figure 1). A unit load (1 lb) is applied at the other end as tip in-plane shear using a rigid body element (RBE2).

### *Solution Type*

Static

### *Comparison of Results*

Tabular results are given in Table 1.



Table 1. Results

| Mesh Size | Element | Bench Value*        | Autodesk Nastran    |           |
|-----------|---------|---------------------|---------------------|-----------|
|           |         | Displacement z (in) | Displacement z (in) | Error (%) |
| 1x9       | CQUAD4  | 5.02                | 4.45                | 11.4      |
|           | CQUADR  | 5.02                | 4.39                | 12.5      |
| 3x17      | CQUAD4  | 5.02                | 4.97                | 1.0       |
|           | CQUADR  | 5.02                | 4.92                | 1.2       |
| 5x34      | CQUAD4  | 5.02                | 5.01                | 0.2       |
|           | CQUADR  | 5.02                | 4.97                | 1.0       |
| 10x68     | CQUAD4  | 5.02                | 5.06                | 0.8       |
|           | CQUADR  | 5.02                | 5.03                | 0.2       |
| 20x136    | CQUAD4  | 5.02                | 5.09                | 1.4       |
|           | CQUADR  | 5.02                | 5.08                | 1.2       |

**\*Note:** The bench value of 5.02 in is cited by Knight (1997) and is based on tests performed using various types of finite elements. An analytical solution for the shell model of the hook is not available. According to Wlassow (1964), the analytical solution is based on the theory of curved beams and it gives a stiffer solution for the deflection, 4.7561 in.

## References

1. Knight, N. F., *The Raasch Challenge for Shell Elements*, AIAA Journal, Volume 35, No.2, 1997. pp. 375-381.
2. Wlassow, W.S., *Dunnwandige Elastische Stabe Band 2*, VEB Verlag fur das Bauwesen Berlin, 1964.

## 2.13 Twisted Beam Static Load

### Problem Description

Figure 1 shows the twisted beam. Static analysis is performed on the plate using two different load cases. The maximum displacement is determined at the tip of the load in the direction of the load in-plane and out-of-plane, respectively. All dimensions are in inches.

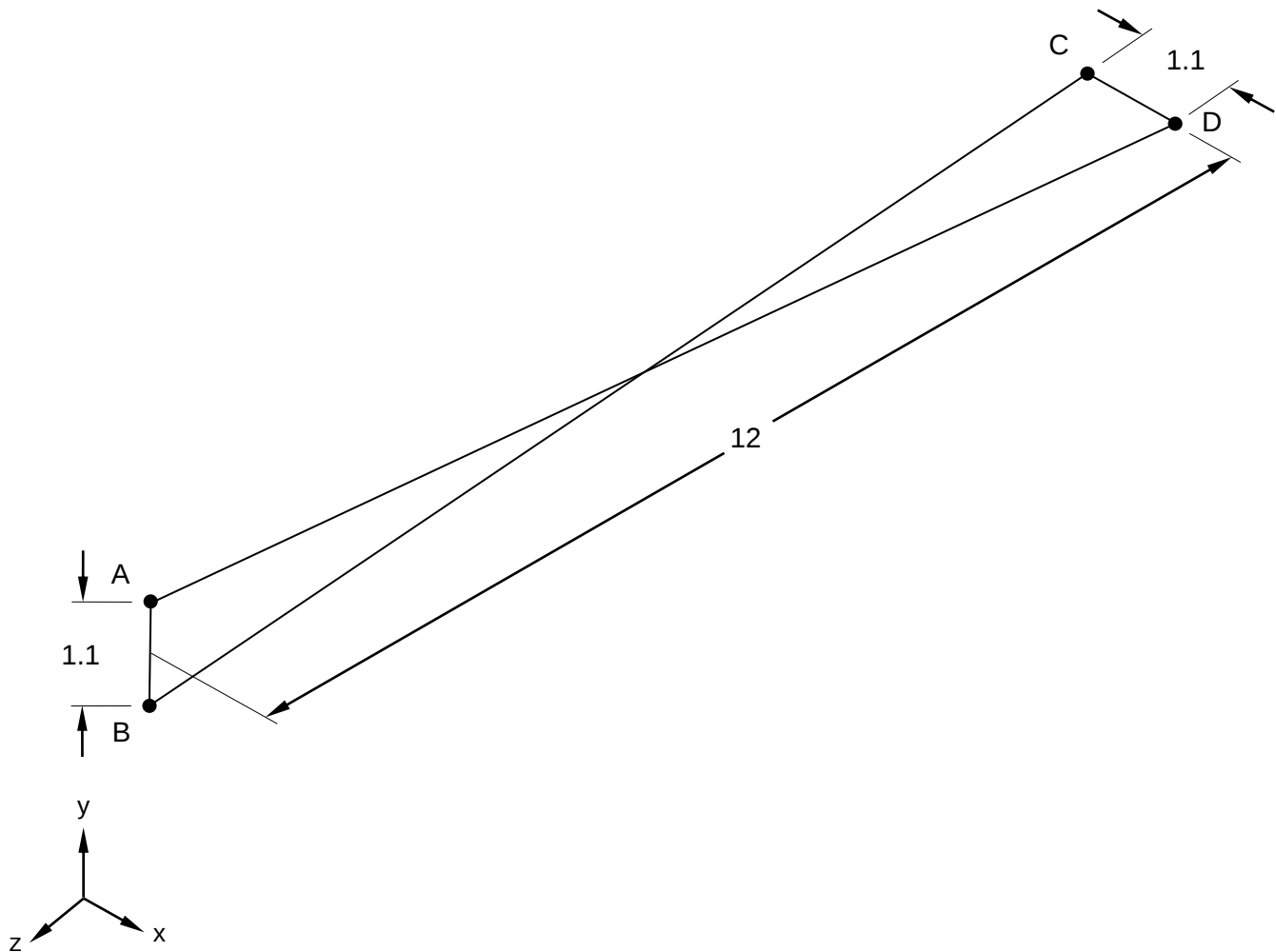


Figure 1. Twisted Beam Model

### Autodesk Nastran Analysis Model Filename

- vm2\_13.nas

### Model Data

#### *Finite Element Modeling*

- 441 nodes, 384 5-DOF/node quadrilateral plate elements

**Units**

inch/pound/second

**Model Geometry**

Length:  $L = 12.00$  in

Width:  $w = 1.10$  in

Thickness:  $t = 0.32$  in

Twist:  $\theta = 90$  (root to tip)

**Material Properties**

Young's Modulus:  $E = 29.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.22$

**Boundary Conditions**

The model is constrained at one end in all translations and all rotations (edge CD). A unit load of 1 lb is applied at tip (edge AB) in-plane (Y-direction) and out-of-plane (in the X-direction), respectively.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                                 | Theory   | Autodesk Nastran | Error (%) |
|-------------------------------------------------------------|----------|------------------|-----------|
| Max Displacement in Y-direction (in) – loading in-plane     | 5.424E-3 | 5.513E-3         | 1.6       |
| Max Displacement in X-direction (in) – loading out-of-plane | 1.754E-3 | 1.779E-3         | 1.4       |

**References**

1. MacNeal, R. H. and Harder, R. C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*. North Holland, 1985.

## 2.14 Plane Frame with Beam Span Loads

### Problem Description

Figure 1 shows the plane frame. Static analysis is performed on the 2D frame using two different load cases. The bending moments and shear forces at mid-point on the first bay for each load case are determined. All dimensions are in feet.

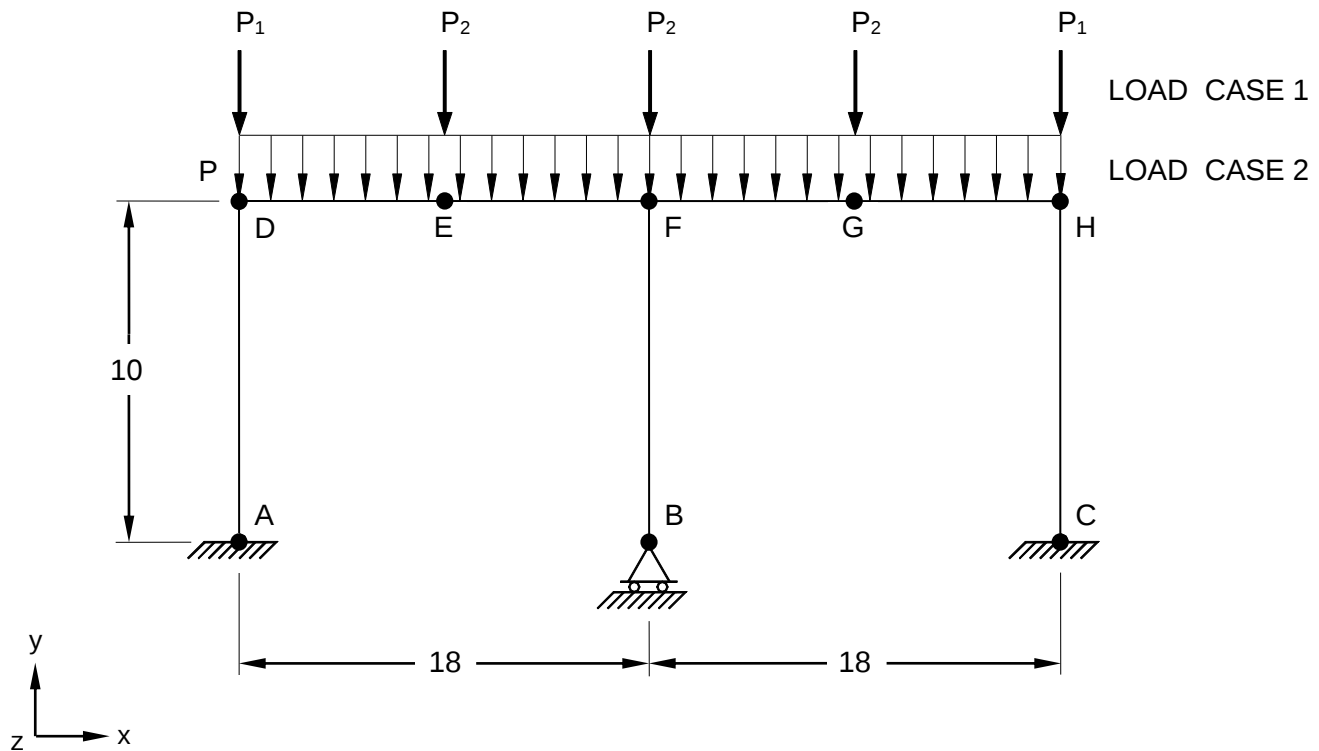


Figure 1. Plane Frame Model with Beam Span Loads

### Autodesk Nastran Analysis Model Filename

- vm2\_14.nas

### Model Data

#### Finite Element Modeling

- 8 nodes, 7 beam elements

#### Units

inch/Kip/second

#### Model Geometry

Length:  $L = 216$  in (18 ft)

Height:  $h = 120$  in (10 ft)

### Cross Sectional Properties

Area:  $A = 1.0E+7 \text{ in}^2$  for both columns and beams  
 Rectangular Cross Section = (12 in x 24 in) for columns  
 Rectangular Cross Section = (12 in x 30 in) for beams  
 Moment of Inertia:  $I_z = 13824 \text{ in}^4$  for columns  
 Moment of Inertia:  $I_z = 27000 \text{ in}^4$  for beams

### Material Properties

Young's Modulus:  $E = 30.0 \text{ E}+3 \text{ ksi}$   
 Shear Modulus of Elasticity:  $G = 15.0 \text{ E}+3 \text{ ksi}$

### Boundary Conditions

The model is constrained at ends A and C in all translations and all rotations. The middle column is pinned at the base, node B (all translations restrained). In addition, the beams at the connection to the left and right columns have the strong moment released. For the first load case, a load P1 of 50 Kip is applied at nodes D and H, and another load P2 of 100 Kip is applied at nodes E, F, and G (negative Y-direction). The second load case is a uniformly distributed load of 10 Kip/ft on both beams, in the negative Y-direction.

### Solution Type

Static

### Comparison of Results

Tabular results are given in Tables 1 and 2.

**Table 1. Test Case 1 Results**

| Description   | Theory | Autodesk Nastran | Error (%) |
|---------------|--------|------------------|-----------|
| Moment (k-in) | 3375   | 3375             | 0.0       |
| Shear (k)     | 68.75  | 68.75            | 0.0       |

**Table 2. Test Case 2 Results**

| Description   | Theory | Autodesk Nastran | Error (%) |
|---------------|--------|------------------|-----------|
| Moment (k-in) | 2430   | 2430             | 0.0       |
| Shear (k)     | 22.50  | 22.50            | 0.2       |

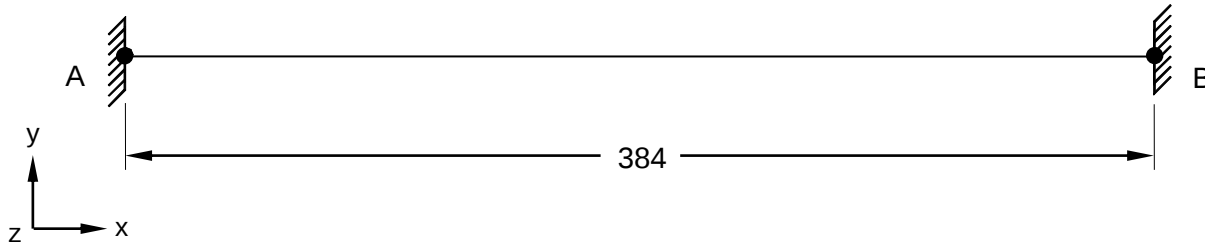
### References

1. *Manual of Steel Construction-Allowable Stress Design*, AISC. Chicago, Illinois, 1989.

## 2.15 Thermal Gradient Loads on a Beam

### Problem Description

Figure 1 shows the cantilever beam. Static analysis is performed on the beam. The bending moment at support and maximum displacement are determined. All dimensions are in inches.



**Figure 1. Beam Model with Thermal Gradient Loads**

### Autodesk Nastran Analysis Model Filename

- vm2\_15.nas

### Model Data

#### *Finite Element Modeling*

- 7 nodes, 6 bar elements

#### *Units*

inch/Kip/second

#### *Model Geometry*

Length:  $L = 384$  in

#### *Cross Sectional Properties*

Area:  $A = 144$  in<sup>2</sup>

Square Cross Section = (12 in x 12 in)

Moment of Inertia:  $I = 1728$  in<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 3150.0$  ksi

Poisson's Ratio:  $\nu = 0.17$

Thermal Expansion Coefficient:  $\alpha = 5.5 \text{ E-06}$  in/in/deg F

**Boundary Conditions**

The model is constrained at both ends in all translations and all rotations. A thermal gradient of 120° F is defined across the depth of the beam for all beams.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description               | Theory | Autodesk Nastran | Error (%) |
|---------------------------|--------|------------------|-----------|
| Maximum Translation (in)  | 0.0    | 0.0              | 0.0       |
| Restraint Moment (Kip-in) | 299.4  | 299.4            | 0.0       |

**References**

1. Roark, R. and Young, W., *Formulas for Stress and Strain*, 6<sup>th</sup> Edition. New York: McGraw-Hill Book Co., 1989. p. 112.

## 2.16 Statically Indeterminate Reaction Force Analysis

### Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The reactions at nodes 1 and 4 (A and D) are determined. All dimensions are in inches.

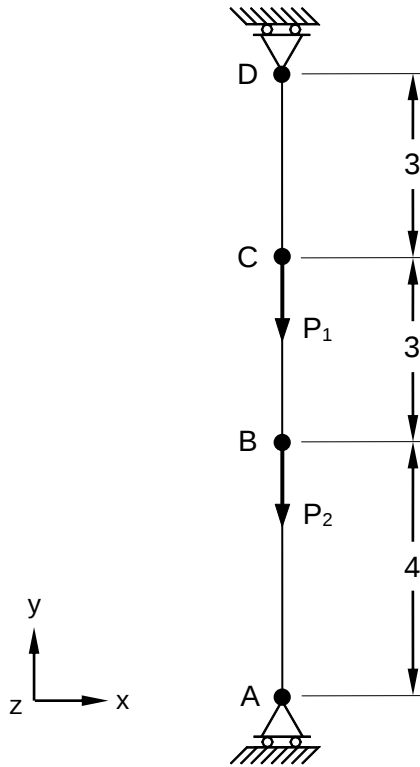


Figure 1. Beam Model

### Autodesk Nastran Analysis Model Filename

- vm2\_16.nas

### Model Data

#### *Finite Element Modeling*

- 4 nodes, 3 bar elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 10$  in



**Cross Sectional Properties**

Area:  $A = 1 \text{ in}^2$

Square Cross Section = (1 in x 1 in)

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+6 \text{ psi}$

Shear Modulus of Elasticity:  $G = 15.0 \text{ E}+6 \text{ psi}$

**Boundary Conditions**

The model is constrained at both ends (A and D) in all translations and rotations in X and Y-directions. A load  $P_1 = 1,000 \text{ lb}$  is applied at node C, and another load  $P_2 = 500 \text{ lb}$  is applied at node B in the negative Y-direction.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description      | Theory | Autodesk Nastran | Error (%) |
|------------------|--------|------------------|-----------|
| Reaction R1 (lb) | 600    | 600              | 0.0       |
| Reaction R2 (lb) | 900    | 900              | 0.0       |

**References**

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955.

## 2.17 Beam Stresses and Deflection

### Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The maximum stress in the middle of the beam and maximum deflection at center of the beam are determined. All dimensions are in inches.

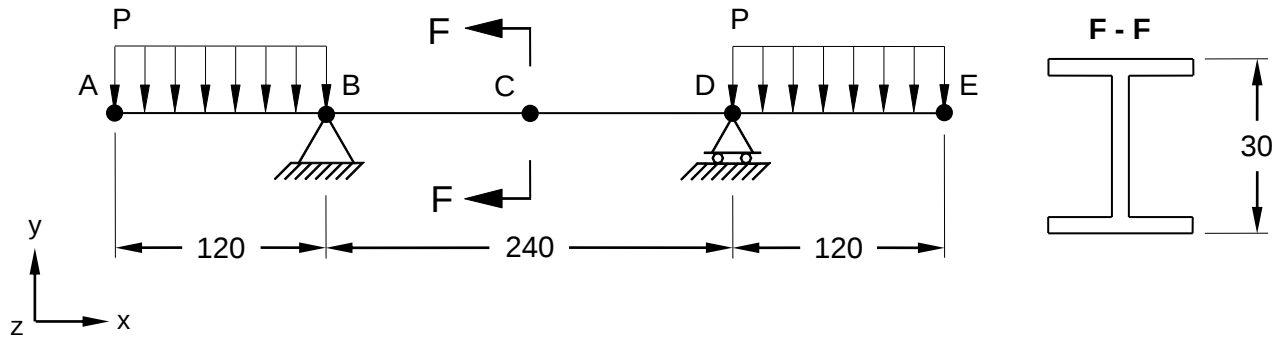


Figure 1. Beam Stresses and Deflection Model

### Autodesk Nastran Analysis Model Filename

- vm2\_17.nas

### Model Data

#### Finite Element Modeling

- 5 nodes, 4 beam elements

#### Units

inch/pound/second

#### Model Geometry

Length:  $L = 480$  in

Height:  $h = 30$  in

#### Cross Sectional Properties

Area:  $A = 50.65$  in<sup>2</sup>

Moment of Inertia:  $I_z = 7892$  in<sup>4</sup>

#### Material Properties

Young's Modulus:  $E = 30.0$  E+6 psi

Shear Modulus of Elasticity:  $G = 15.0$  E+6 psi

**Boundary Conditions**

One joint is pinned (at point B) and the other one is a roller (at point D). Points A, C and E are constrained in the Z-translation, and X and Y-rotations. A uniform distributed load  $P = 833.33 \text{ lb/in}$  is applied on each cantilever in the negative Y-direction.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                  | Theory | Autodesk Nastran | Error (%) |
|------------------------------|--------|------------------|-----------|
| Maximum Bending Stress (psi) | 11,404 | 11,404           | 0.0       |
| Maximum Deflection (in)      | 0.182  | 0.182            | 0.0       |

**References**

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955.

## 2.18 Laterally Loaded Tapered Support Structure Case 1

### Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The maximum bending stress at mid span (point B) is determined. All dimensions are in inches.

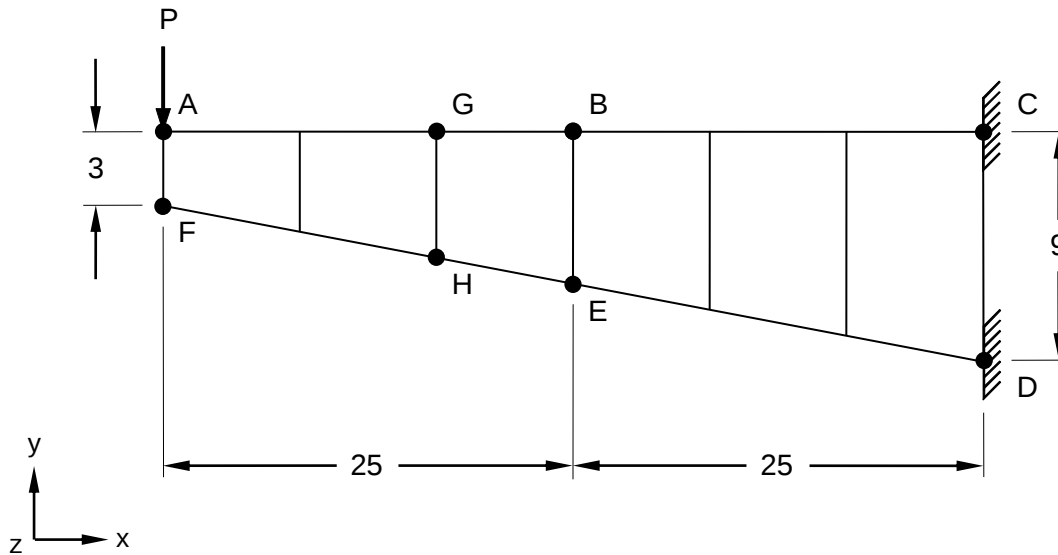


Figure 1. Tapered Support Structure Model Case 1

### Autodesk Nastran Analysis Model Filename

- vm2\_18.nas

### Model Data

#### *Finite Element Modeling*

- 14 nodes, 6 6-DOF/node quadrilateral plate elements, and 2 beam elements along edges BG and EH (used to recover the stress at mid-span)

### Units

inch/pound/second

### Model Geometry

Length:  $L = 50$  in

Width:  $w = 2$  in

Height:  $h_1 = 3$  in and  $h_2 = 9$  in

### Material Properties

Young's Modulus:  $E = 30.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.2$

**Boundary Conditions**

The deep end (CD) is fixed in all translations and all rotations. All the other nodes are constrained in the Z-translation and X and Y-rotations. A load  $P = 4,000$  lb is applied at the free end of the cantilever in the negative Y-direction.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                              | Theory | Autodesk Nastran | Error (%) |
|------------------------------------------|--------|------------------|-----------|
| Maximum Bending Stress at Mid-Span (psi) | 8,333  | 8,234            | 1.2       |

**References**

1. Crandall, S. H. and Dahl, N. C., *An Introduction to the Mechanics of Solids*, 3<sup>rd</sup> Edition. New York: McGraw-Hill, Inc., 1959.

## 2.19 Laterally Loaded Tapered Support Structure Case 2

### Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The maximum bending stress at mid span (point B) is determined. All dimensions are in inches.

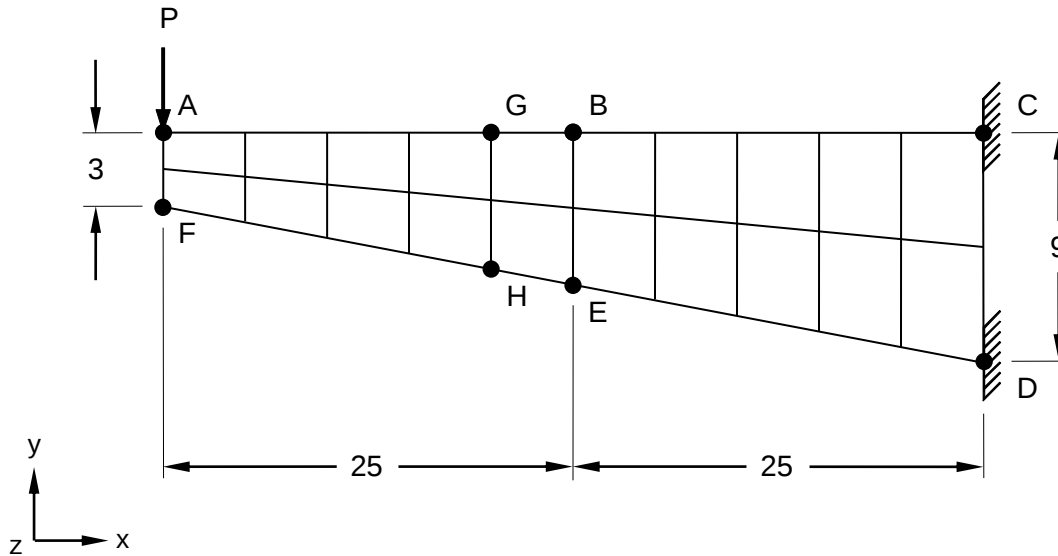


Figure 1. Tapered Support Structure Model Case 2

### Autodesk Nastran Analysis Model Filename

- vm2\_19.nas

### Model Data

#### *Finite Element Modeling*

- 33 nodes, 20 6-DOF/node quadrilateral plate elements, and 2 beam elements along edges BG and EH (used to recover the stress at mid-span)

### Units

inch/pound/second

### Model Geometry

Length:  $L = 50$  in

Width:  $w = 2$  in

Height:  $h_1 = 3$  in and  $h_2 = 9$  in

### Material Properties

Young's Modulus:  $E = 30.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.2$

**Boundary Conditions**

The deep end (CD) is fixed in all translations and all rotations. All the other nodes are constrained in the Z-translation and X and Y-rotations. A load  $P = 4,000$  lb is applied at the free end of the cantilever in the negative Y-direction.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                              | Theory | Autodesk Nastran | Error (%) |
|------------------------------------------|--------|------------------|-----------|
| Maximum Bending Stress at Mid-Span (psi) | 8,333  | 8,330            | 0.0       |

**References**

1. Crandall, S. H. and Dahl, N. C., *An Introduction to the Mechanics of Solids*, 3<sup>rd</sup> Edition. New York: McGraw-Hill, Inc., 1959.

## 2.20 Bending of a Tee-Shaped Beam

### Problem Description

Figure 1 shows the short tee-shaped beam model and a cross-section of the beam. Static analysis is performed on the beam. The maximum tensile and compressive stresses are determined. All dimensions are in inches.

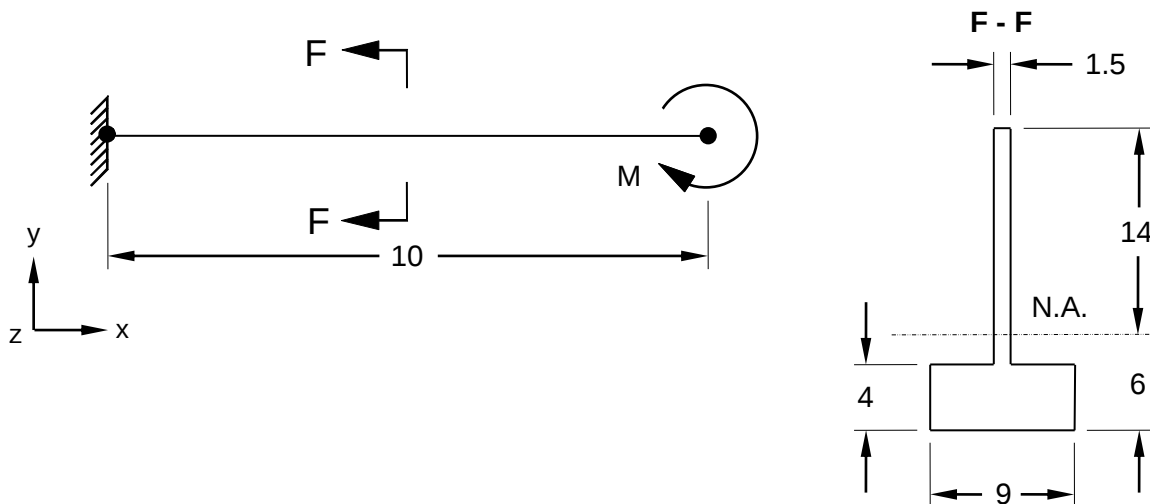


Figure 1. Tee-Shaped Beam Model

### Autodesk Nastran Analysis Model Filename

- vm2\_20.nas

### Model Data

#### Finite Element Modeling

- 2 nodes, 1 beam element

#### Units

inch/pound/second

#### Model Geometry

Length:  $L = 10$  in

Width:  $w = 9$  in

Height to Neutral Axis:  $h_1 = 6$  in

Height:  $h_2 = 20$  in

Thickness Flange:  $t_f = 4$  in

Thickness Web:  $t_w = 1.5$  in

#### Cross Sectional Properties

Area:  $A = 60$  in<sup>2</sup>

Moment of Inertia:  $I_z = 2000$  in<sup>4</sup>



**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+6 \text{ psi}$

Shear Modulus of Elasticity:  $G = 15.0 \text{ E}+6 \text{ psi}$

**Boundary Conditions**

The left end is fixed in all translations and all rotations. The right end is constrained in the Z-translation and X and Y-rotations. A moment  $M = 100,000 \text{ lb-in}$  is applied at the free end of the cantilever in the positive Z-direction.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                      | Theory | Autodesk Nastran | Error (%) |
|----------------------------------|--------|------------------|-----------|
| Maximum Tensile Stress (psi)     | 300    | 300              | 0.0       |
| Maximum Compressive Stress (psi) | -700   | -700             | 0.0       |

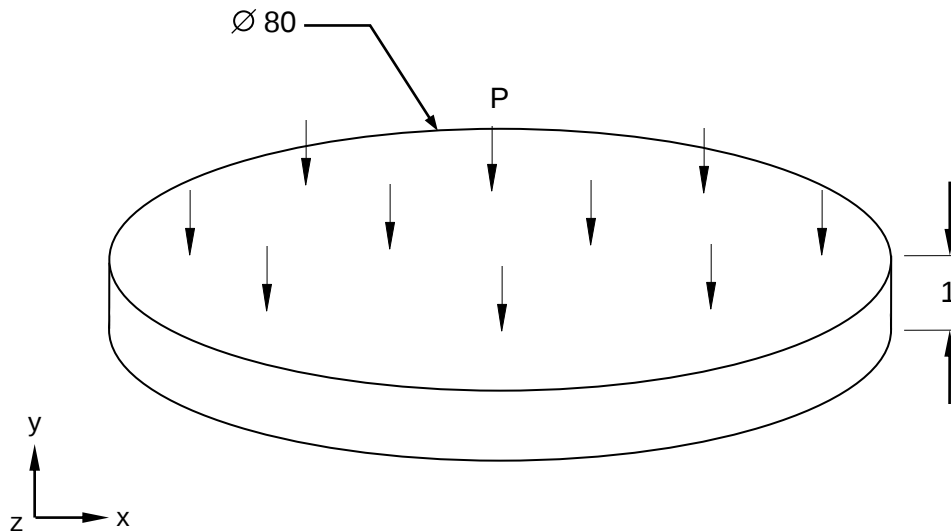
**References**

1. Crandall, S. H. and Dahl, N. C., *An Introduction to the Mechanics of Solids*, 3<sup>rd</sup> Edition. New York: McGraw-Hill, Inc., 1959

## 2.21 Bending of a Circular Plate

### Problem Description

Figure 1 shows the plate model with pressure loading. Static analysis is performed on the plate. The maximum deflection in the middle of the plate and the maximum stress in the plate are determined. All dimensions are in inches.



**Figure 1. Circular Plate Model**

### Autodesk Nastran Analysis Model Filename

- vm2\_21.nas

### Model Data

#### *Finite Element Modeling*

- 641 nodes, 640 5-DOF/node quadrilateral plate elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Radius:  $R = 40$  in

Thickness:  $t = 1$  in

#### *Material Properties*

Young's Modulus:  $E = 30.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

The outside edge is fixed in all translations and all rotations. A uniform pressure  $P = 6$  psi is applied to the entire surface of the plate.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description               | Theory    | Autodesk Nastran | Error (%) |
|---------------------------|-----------|------------------|-----------|
| Maximum Displacement (in) | -8.736E-2 | -8.603E-2        | 1.5       |
| Maximum Stress (psi)      | 7200      | 7164             | 0.5       |

**References**

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1956.

## 2.22 Beam on Elastic Foundation

### Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The maximum deflection and bending stress at the center of the beam are determined. All dimensions are in inches.

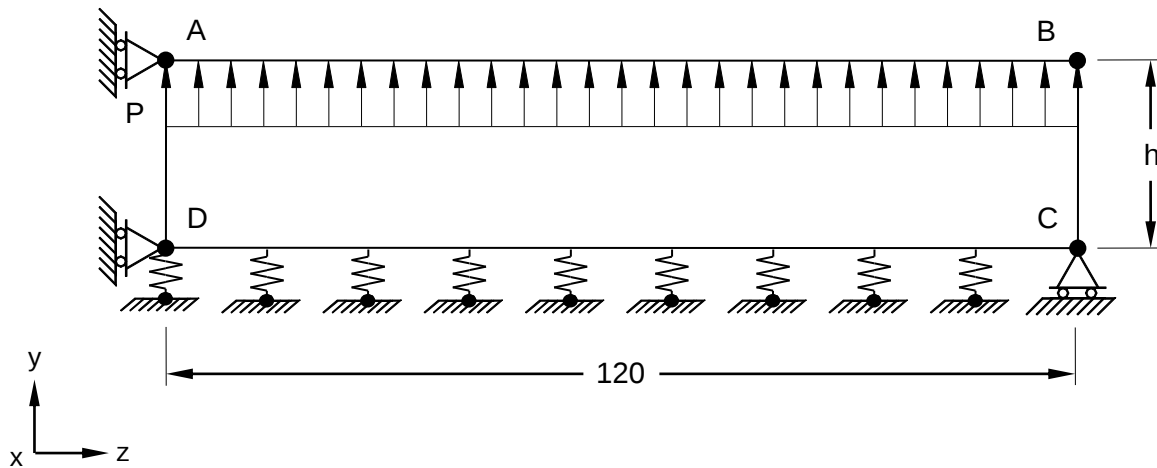


Figure 1. Beam Model on Elastic Foundation

### Autodesk Nastran Analysis Model Filename

- vm2\_22.nas

### Model Data

#### Finite Element Modeling

- A 1/2 model is created with 62 nodes, 20 5-DOF/node quadrilateral plate elements, 20 rod elements for the springs, and 20 beam elements placed along the top of the plates (used to recover the stress at mid-span).

### Units

inch/pound/second

### Model Geometry

Length:  $L = 240$  in

Height:  $H = 7.113786$  in

Thickness:  $t = 1$  in

### Material Properties

Young's Modulus:  $E = 30.0 \text{ E}+6$  psi

Shear Modulus of Elasticity:  $G = 15.0 \text{ E}+6$  psi

Spring Constant:  $k = 156.25 \text{ lb/in}$  ( $k = 78.125 \text{ lb/in}$  at mid point due to symmetry)

### Boundary Conditions

The end, node C, is constrained in the X and Y-translations, and in the Y and Z-rotations (roller). Springs are attached at each node along the bottom edge DC, except at node C. At mid symmetry, top and bottom nodes (A and D) are restrained against movement in the horizontal translation (Z-direction) in addition to one half of the spring constant in the vertical direction. A uniform load  $P = 43.4$  lbs/in is applied to the entire top part of the beam in the positive Y-direction.

### Solution Type

Static

### Comparison of Results

Tabular results are given in Table 1.

**Table 1. Results**

| Description                  | Theory | Autodesk Nastran | Error (%) |
|------------------------------|--------|------------------|-----------|
| Maximum Displacement (in)    | 1.0453 | 1.0455           | 0.0       |
| Maximum Bending Stress (psi) | 18052  | 18058            | 0.0       |

### References

1. Peterson, F. E., *EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual*, Engineering Analysis Corporation. Berkeley, California, 1981.

## 2.23 Thick Walled Cylinder Plain Strain

### Problem Description

Figure 1 shows the thick walled cylinder model. Static analysis is performed on the beam. The radial displacements, radial stress, tangential and longitudinal stresses at inner surface of the thick wall cylinder are determined. All dimensions are in inches.

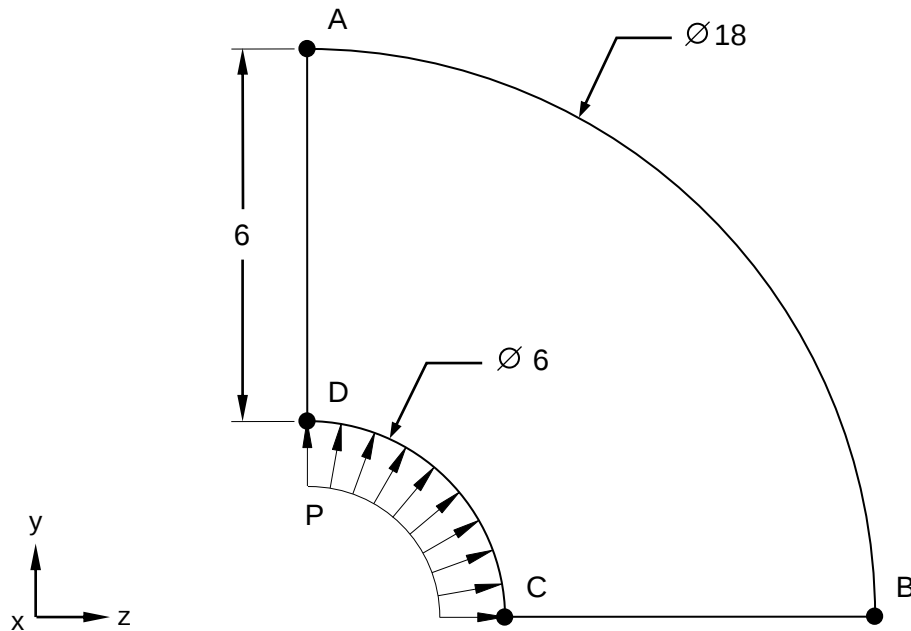


Figure 1. Thick Walled Cylinder Model

### Autodesk Nastran Analysis Model Filename

- vm2\_23.nas

### Model Data

#### Finite Element Modeling

- A  $\frac{1}{4}$  symmetry model is created using joint restraints with 589 nodes, 540 6-DOF/node quadrilateral plate elements, and 18 beam elements placed along the inner circle (used for load distribution).

#### Units

inch/pound/second

#### Model Geometry

Inner Radius:  $R_i = 3$  in  
 Outer Radius:  $R_o = 9$  in  
 Thickness:  $t = 1$  in

**Material Properties**

Young's Modulus:  $E = 1.0 \text{ E}+6 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All nodes are constrained against movement in the direction perpendicular to the surface (translation in the Z-direction and rotations in the X and Y-directions). Symmetry boundary conditions are applied to the AD and BC edges. A unit pressure load  $P = 1 \text{ psi}$  is applied on the inner radius on the beams.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                    | Theory | Autodesk Nastran | Error (%) |
|--------------------------------|--------|------------------|-----------|
| Radial Displacement x1E-5 (in) | 0.4582 | 0.4649           | 1.5       |
| Radial Stress (psi)            | -1.0   | -0.98            | 2.0       |
| Tangential Stress (psi)        | 1.25   | 1.25             | 0.0       |

**References**

1. McNeal, R. H. and Harder, R. C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*. North Holland, 1985.

## 2.24 Scordelis-Lo Roof

### Problem Description

Figure 1 shows the roof model. Static analysis is performed on the roof. The vertical displacement at the center of the free edge, top and bottom stresses at the centerline section at vertical angle, and top and bottom stresses at the centerline section at the free edge are determined. All dimensions are in feet.

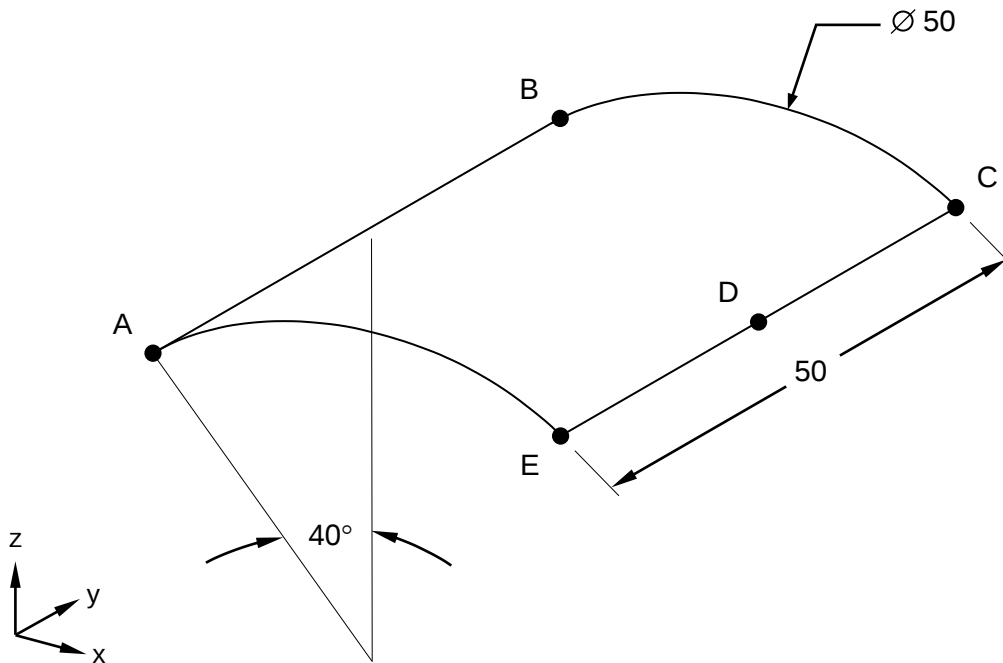


Figure 1. Scordelis-Lo Roof Model

### Autodesk Nastran Analysis Model Filename

- vm2\_24.nas

### Model Data

#### Finite Element Modeling

- A  $\frac{1}{2}$  symmetry model is created using 1387 nodes and 1296 6-DOF/node quadrilateral plate elements.

### Units

foot/pound/second

### Model Geometry

Length:  $L = 50$  feet

Radius:  $R = 25$  feet

Thickness:  $t = 0.25$  feet



### Material Properties

Young's Modulus:  $E = 4.32 \text{ E}+8 \text{ psf}$

Poisson's Ratio:  $\nu = 0.17$

### Boundary Conditions

All nodes on the curved edges are simply supported (constrained translation in the X and Z-directions and rotation in the Y-direction). Symmetry boundary conditions are applied to the AB edge. The node D at the centerline section on the free edge (CE) is constrained in the Y-translation to stabilize the model. A gravity load  $P = 90 \text{ psf}$  is applied uniformly on the surface.

### Solution Type

Static

### Comparison of Results

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                                          | Theory  | Autodesk Nastran | Error (%) |
|----------------------------------------------------------------------|---------|------------------|-----------|
| Displacement at Center of Free Edge (ft) – Vertical                  | -0.3019 | -0.3152          | 4.4       |
| Displacement at Center of Free Edge (ft) – Horizontal                | -0.1593 | -0.1673          | 5.0       |
| Stress at Centerline Section at Vertical Angle x 1E+3 (ksf) – Top    | 191.23  | 195.53           | 2.2       |
| Stress at Centerline Section at Vertical Angle x 1E+3 (ksf) – Bottom | -218.74 | -222.64          | 1.8       |
| Stress at Centerline Section at Free End x 1E+3 (ksf) – Top          | 215.57  | 242.19           | 12.3      |
| Stress at Centerline Section at Free End x 1E+3 (ksf) – Bottom       | 340.70  | 374.19           | 9.8       |

### References

1. Scordelis, A. C. and Lo, K. S., *Computer Analysis of Cylindrical Shells*, Journal of the American Concrete Institute, Vol. 61, May 1964.

## 2.25 Out-of-Plane Bending of a Curved Bar

### Problem Description

Figure 1 shows the curved bar model (90 degrees arc). Static analysis is performed on the bar. The maximum displacement at the free end, and maximum bending stress at support are determined. All dimensions are in inches.

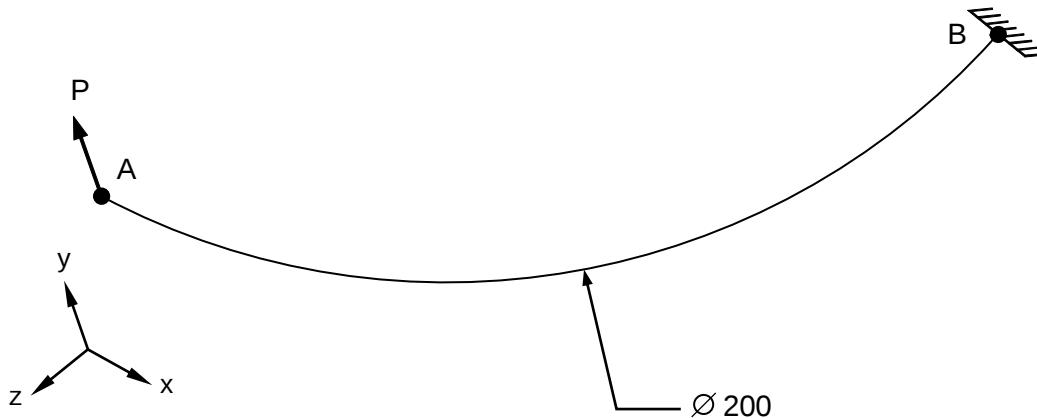


Figure 1. Curved Bar Model

### Autodesk Nastran Analysis Model Filename

- vm2\_25.nas

### Model Data

#### *Finite Element Modeling*

- 19 nodes, 18 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Arc: 90 degrees

Arc Radius:  $R = 100$  in

Bar Radius:  $R_{bar} = 1$  in

#### *Material Properties*

Young's Modulus:  $E = 30.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

One end (point B) is fixed in all translations and rotations. A load  $P = 50$  lb is applied at the free end (point A), out of plane to the curved beam.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                             | Theory | Autodesk Nastran | Error (%) |
|-----------------------------------------|--------|------------------|-----------|
| Displacement at Free End (in)           | -2.648 | -2.647           | 0.0       |
| Maximum Bending Stress at Support (psi) | 6366   | 6638             | 4.3       |

**References**

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955.

## 2.26 Deflection of Hinged Support

### Problem Description

Figure 1 shows the hinged beam model. Static analysis is performed on the beams. The maximum displacement at point C (node 2) and stress in each beam are determined. All dimensions are in inches.

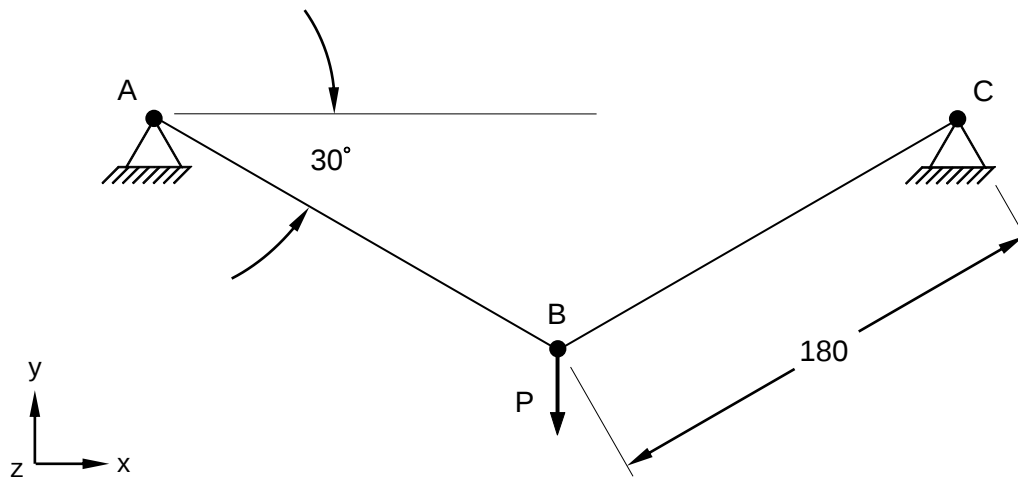


Figure 1. Hinged Support Model

### Autodesk Nastran Analysis Model Filename

- vm2\_26.nas

### Model Data

#### *Finite Element Modeling*

- 3 nodes, 2 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 180$  in

Angle:  $\alpha = 30$  degrees

Beam Radius:  $R_{beam} = 1$  in

#### *Cross Sectional Properties*

Area:  $A = 0.50$  in<sup>2</sup>

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+6 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All translations and X and Y-rotations are restrained for points A and B. A point load  $P = 5000 \text{ lb}$  is applied at point C (node 2) in the negative Y-direction.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                 | Theory | Autodesk Nastran | Error (%) |
|-----------------------------|--------|------------------|-----------|
| Displacement at Node 2 (in) | -0.12  | -0.12            | 0.0       |
| Maximum Stress (psi)        | 10,000 | 10,000           | 0.0       |

**References**

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955.

## 2.27 Thermal Stresses in a Plate

### Problem Description

Figure 1 shows the plate model. Static analysis is performed on the plate. The variation in temperature from the inside to the outside follows a linear law. The bending moment per unit length of the clamped edge to prevent the plate from bending and the maximum bending stress are determined. All dimensions are in inches.

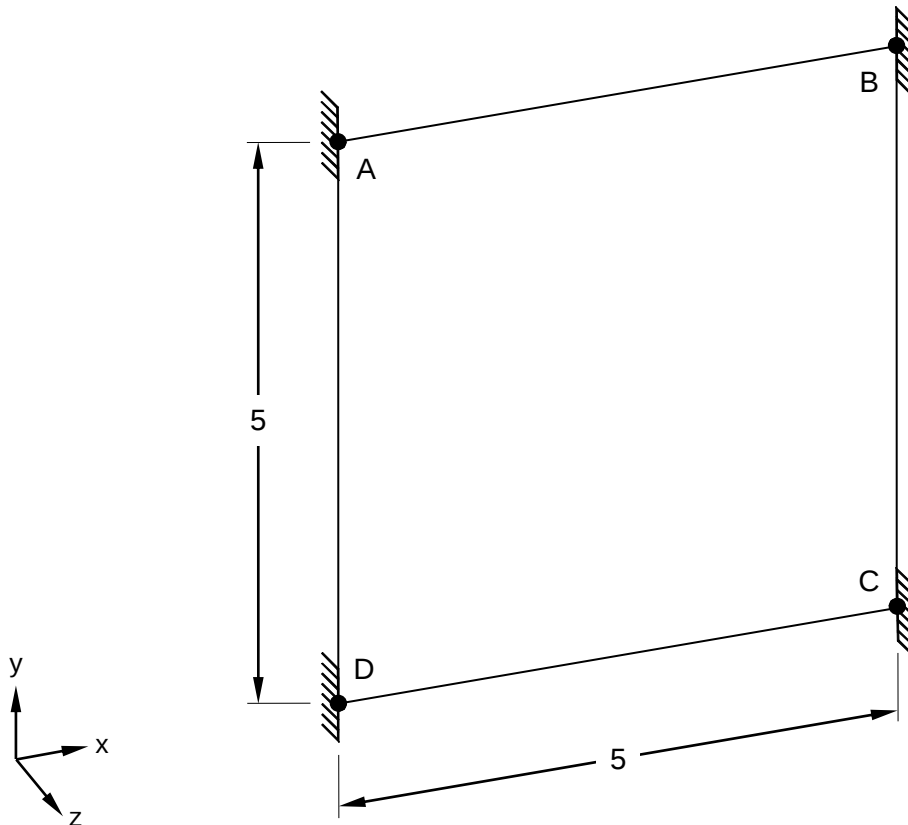


Figure 1. Plate Model

### Autodesk Nastran Analysis Model Filename

- vm2\_27.nas

### Model Data

#### *Finite Element Modeling*

- 4 nodes, 1 5-DOF/node quadrilateral element

#### *Units*

inch/pound/second

**Model Geometry**

Length:  $L = 5$  in

Thickness:  $t = 0.5$  in

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.3$

Thermal Expansion Coefficient:  $\alpha = 7.0 \text{ E}-6$  in/in- °F

Temperature one Face:  $T_0 = 0$  °F

Temperature the other Face:  $T_1 = 100$  °F

**Boundary Conditions**

All translations and rotations are restrained for points A, B, C and D (clamped edges). A temperature load from one face to the other of 100° F is applied to the plate. The temperature varies linearly from one face to another.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description               | Theory  | Autodesk Nastran | Error (%) |
|---------------------------|---------|------------------|-----------|
| Bending Moment (in-lb/in) | -625    | -625             | 0.0       |
| Maximum Stress (psi)      | -15,000 | -15,000          | 0.0       |

**References**

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1956.

## 2.28 Bending of a Tapered Plate

### Problem Description

Figure 1 shows the tapered plate model. Static analysis is performed on the plate. The maximum deflection and maximum principal stress in the plate are determined. All dimensions are in inches.

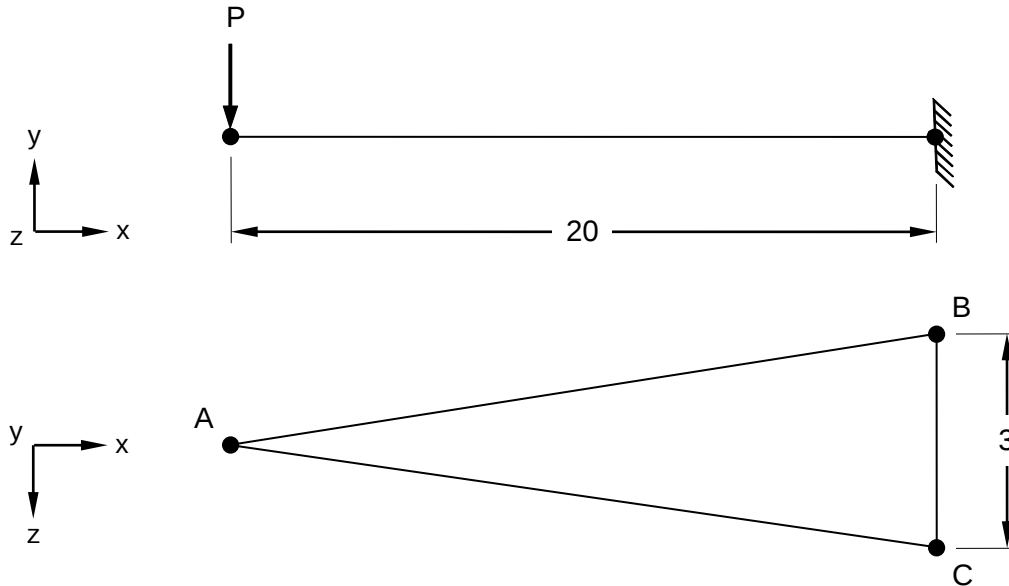


Figure 1. Tapered Plate Model

### Autodesk Nastran Analysis Model Filename

- vm2\_28.nas

### Model Data

#### *Finite Element Modeling*

- 41 nodes, 38 5-DOF/node triangle elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 20$  in

Width:  $w = 3$  in

Thickness:  $t = 0.5$  in

#### *Material Properties*

Young's Modulus:  $E = 30.0 \text{ E}+6$  psi

Poisson's Ratio:  $\nu = 0.3$



**Boundary Conditions**

All translations and rotations are restrained for points B and C. A vertical point load  $P = 10$  lb is applied to the plate model at point A (see Figure 1).

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                    | Theory     | Autodesk Nastran | Error (%) |
|--------------------------------|------------|------------------|-----------|
| Displacement (in)              | -4.2667E-2 | -4.2556E-2       | 0.3       |
| Maximum Principal Stress (psi) | 1,600      | 1,600            | 0.0       |

**References**

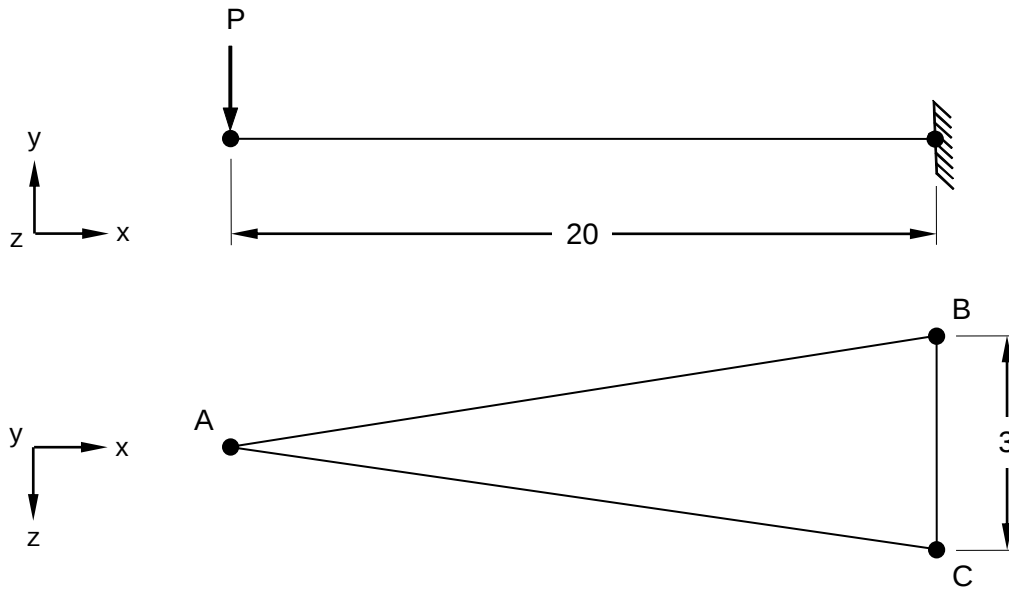
1. Harris, C. O., *Introduction to Stress Analysis*. New York: The Macmillan Co., 1959.

## 2.29 Bending of a Tapered Beam

### Problem Description

Figure 1 shows the tapered beam model. Static analysis is performed on the beam. The maximum deflection and maximum principal stress in the beam are determined. All dimensions are in inches.

This problem is used to compare the results to that of a finite element plate model (see problem 2.28). The results compare favorably.



**Figure 1. Tapered Beam Model**

### Autodesk Nastran Analysis Model Filename

- vm2\_29.nas

### Model Data

#### *Finite Element Modeling*

- 11 nodes, 10 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 20$  in

Width:  $w = 3$  in

Thickness:  $t = 0.5$  in

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+6 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All translations and rotations are restrained for points B and C. A vertical point load  $P = 10 \text{ lb}$  is applied to the beam model at point A (see Figure 1).

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                    | Theory     | Autodesk Nastran | Error (%) |
|--------------------------------|------------|------------------|-----------|
| Displacement (in)              | -4.2667E-2 | -4.2667E-2       | 0.0       |
| Maximum Principal Stress (psi) | 1.600E+3   | 1.613E+3         | 0.8       |

**References**

1. Harris, C. O., *Introduction to Stress Analysis*. New York: The Macmillan Co., 1959.

## 2.30 Bending of a Curved Thick Beam

### Problem Description

Figure 1 shows the curved thick beam model. Static analysis is performed on the beam. The maximum tensile stress and maximum compressive stress in the beam are determined. All dimensions are in inches.

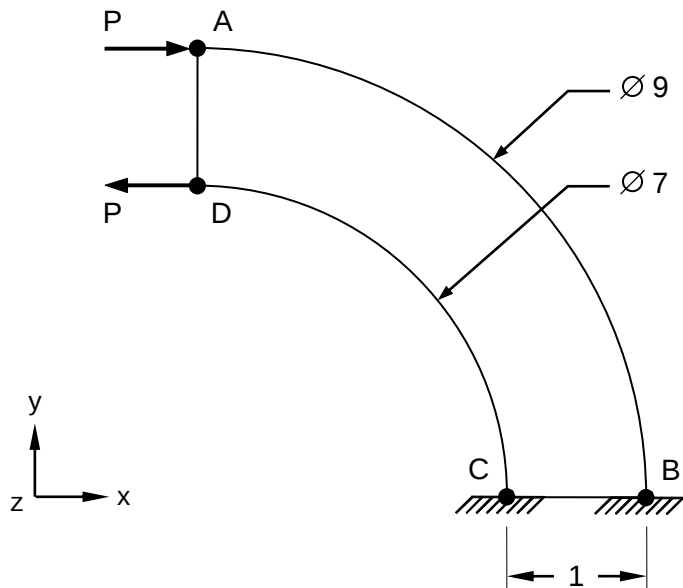


Figure 1. Curved Thick Beam Model

### Autodesk Nastran Analysis Model Filename

- vm2\_30.nas

### Model Data

#### Finite Element Modeling

- A  $\frac{1}{4}$  model (90 degrees arc) model is created using 105 nodes, 80 5-DOF/node quadrilateral elements and 4 beam elements on the edge AD. Symmetry boundary conditions are applied.

#### Units

inch/pound/second

#### Model Geometry

Inner Radius:  $R_i = 3.5$  in

Outer Radius:  $R_o = 4.5$  in

Width:  $w = 1$  in

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+6 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All translations and rotations are restrained for all nodes between points B and C. A force couple  $P = 100 \text{ lb}$  is applied at the free end of the beam to simulate a 100 in-lb moment (see Figure 1).

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description              | Theory | Autodesk Nastran | Error (%) |
|--------------------------|--------|------------------|-----------|
| Tensile Stress (psi)     | 655.0  | 655.5            | 0.1       |
| Compressive Stress (psi) | -555.0 | -552.1           | 0.5       |

**References**

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955. p.98.

## 2.31 Truss Reaction One

### Problem Description

Figure 1 shows the truss model. Static analysis is performed on the truss. The vertical reaction at the supports of the truss are determined. All dimensions are in inches.

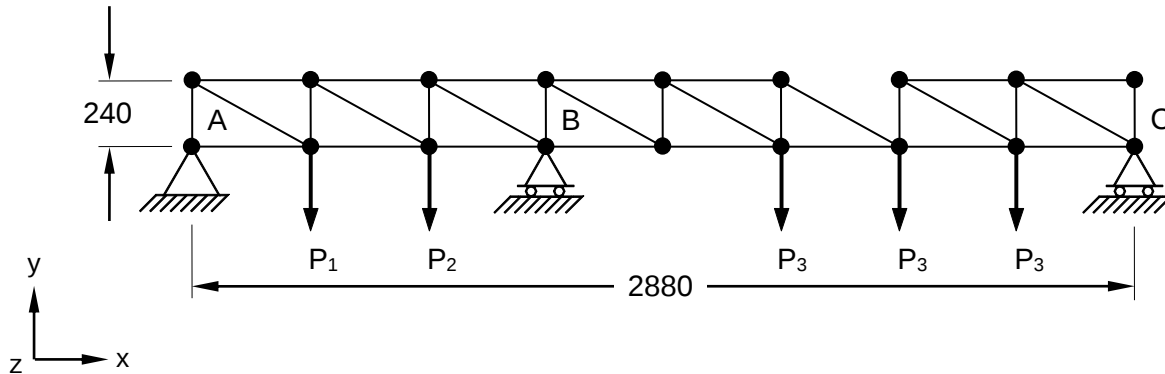


Figure 1. Truss Model

### Autodesk Nastran Analysis Model Filename

- vm2\_31.nas

### Model Data

#### *Finite Element Modeling*

- 18 nodes, 32 beam elements

#### *Units*

inch/Kip/second

#### *Model Geometry*

Length:  $L = 2880$  in (8x360 in)

Height:  $h = 240$  in

#### *Cross Sectional Properties*

Area:  $A = 100$  in<sup>2</sup> (for input only)

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+3 ksi

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

All translations are restrained at point A (pinned support). At points B and C, translations in the Y and Z-directions, and rotations in the X and Y-directions are restrained (rollers). All nodes are constrained in the Z-translation, X and Y-rotations. Several point loads are applied as shown in Figure 1:  $P_1 = 40$  Kip,  $P_2 = 80$  Kip, and  $P_3 = 60$  Kip.

### Solution Type

Static

### Comparison of Results

Tabular results are given in Table 1.

**Table 1. Results**

| Description               | Theory | Autodesk Nastran | Error (%) |
|---------------------------|--------|------------------|-----------|
| Reaction at Point A (Kip) | -76.7  | -76.7            | 0.0       |
| Reaction at Point B (Kip) | 346.7  | 346.7            | 0.0       |
| Reaction at Point C (Kip) | 30.0   | 30.0             | 0.0       |

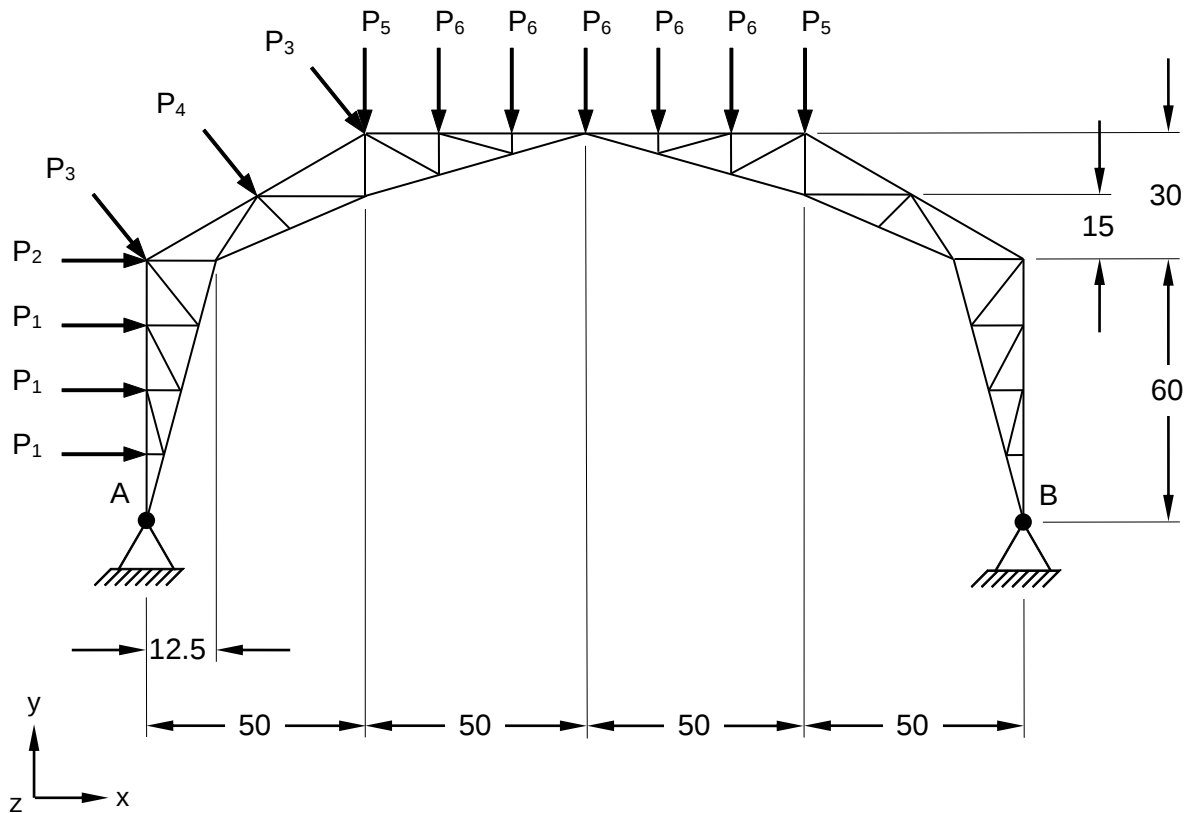
### References

1. McCormac, J. C., *Structural Analysis*, 3<sup>rd</sup> Edition. New York: In Text Educational Publishers, 1975.

## 2.32 Truss Reaction Two

## Problem Description

Figure 1 shows the truss model. Static analysis is performed on the truss. The vertical reaction at the supports of the truss are determined. All dimensions are in inches.



### Figure 1. Truss Model

## Autodesk Nastran Analysis Model Filename

- vm2\_32.nas

## Model Data

## Finite Element Modeling

- 35 nodes, 66 beam elements

## Units

inch/Kip/second



**Model Geometry**

Length:  $L = 200$  in

Height:  $h = 90$  in

**Cross Sectional Properties**

Area:  $A = 100$  in<sup>2</sup> (for input only)

**Material Properties**

Young's Modulus:  $E = 30.0$  E+3 ksi

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All translations are restrained at points A and B (pinned support). All nodes are constrained in the Z-translation, X and Y-rotations. Several point loads are applied as shown in Figure 1:  $P_1 = 4$  Kip,  $P_2 = 2$  Kip,  $P_3 = 3$  Kip,  $P_4 = 6$  Kip,  $P_5 = 5$  Kip, and  $P_6 = 10$  Kip.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                          | Theory | Autodesk Nastran | Error (%) |
|--------------------------------------|--------|------------------|-----------|
| Vertical Reaction at Point A (Kip)   | 34.3   | 34.3             | 0.0       |
| Horizontal Reaction at Point A (Kip) | 11.5   | 11.5             | 0.0       |
| Vertical Reaction at Point B (Kip)   | 36.0   | 36.0             | 0.0       |
| Horizontal Reaction at Point B (Kip) | -31.7  | -31.7            | 0.0       |

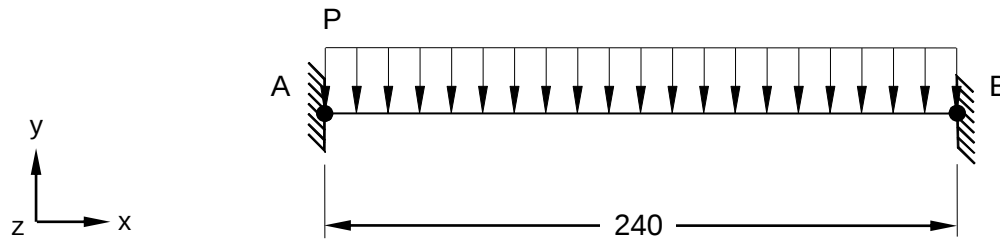
**References**

1. McCormac, J. C., *Structural Analysis*, 3<sup>rd</sup> Edition. New York: In Text Educational Publishers, 1975.

## 2.33 Fixed Ended Beam Un-symmetric Tapered Member

### Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The moments at each of the supports are determined. All dimensions are in inches.



**Figure 1. Beam Model**

### Autodesk Nastran Analysis Model Filename

- vm2\_33.nas

### Model Data

#### *Finite Element Modeling*

- 3 nodes, 2 beam elements

#### *Units*

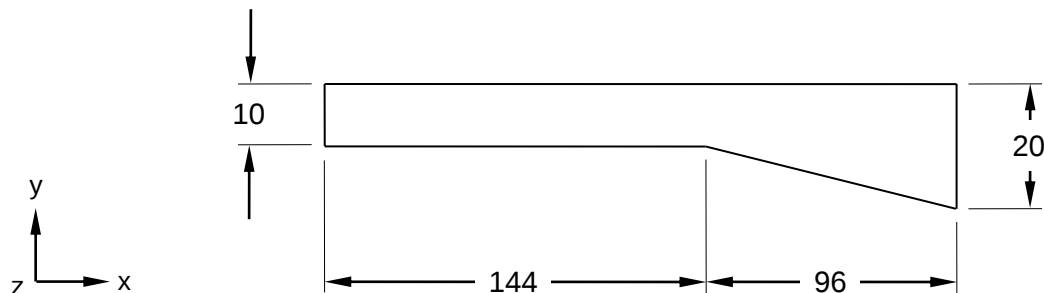
inch/Kip/second

#### *Model Geometry*

Length:  $L = 240$  in

Height:  $h_1 = 10$  in (constant section) and  $h_2 = 20$  in (the deep end of the variable section)

Width:  $w = 1$  in



**Figure 2. Un-symmetric Tapered Beam**

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+3 \text{ ksi}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All translations and rotations are restrained at points A and B (see Figure 1). A uniform load  $P = 4 \text{ Kip/ft}$  (or  $0.3333 \text{ Kip/inch}$ ) is applied to the entire beam.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description             | Theory | Autodesk Nastran | Error (%) |
|-------------------------|--------|------------------|-----------|
| Moment at Point A (Kip) | -98.2  | -95.9            | 2.3       |
| Moment at Point B (Kip) | -217.2 | -223.5           | 2.9       |

**References**

1. McCormac, J. C., *Structural Analysis*, 3<sup>rd</sup> Edition. New York: In Text Educational Publishers, 1975.

## 2.34 Straight Cantilever Beam Using Solids

### Problem Description

Figure 1 shows the straight beam model. Static analysis is performed on the cantilever beam. The displacements in the direction of the loads are determined. All dimensions are in inches.

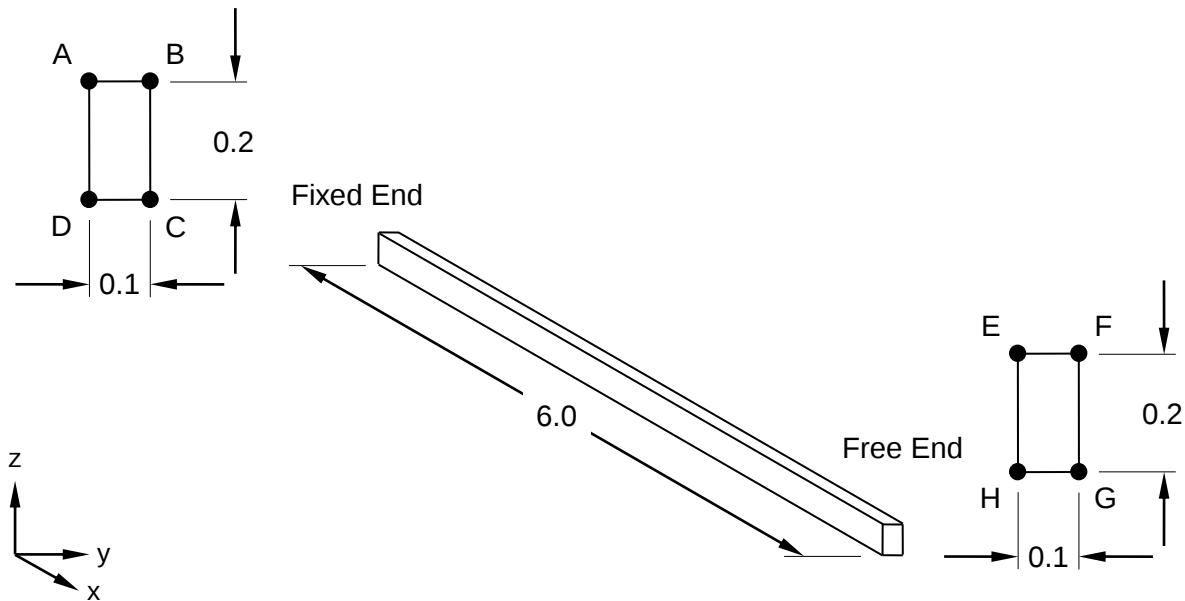


Figure 1. Beam Model

### Autodesk Nastran Analysis Model Filename

- vm2\_34.nas

### Model Data

#### Finite Element Modeling

- 28 nodes, 6 solid elements

#### Units

inch/Kip/second

#### Model Geometry

Length:  $L = 6.0$  in

Height:  $h = 0.2$  in

Width:  $w = 0.1$  in

#### Material Properties

Young's Modulus:  $E = 1.0 \text{ E}+7$  ksi

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

At points A and B the nodes are restrained in the X-translation, while at points C and D the nodes are restrained in all translations (pinned). A unit axial load  $P$  in the X-direction, a unit load at tip along Z-direction, a unit load at tip along Y-direction, and a unit couple at tip about the X-axis via force couple  $P = 5$  Kip along edge HG and edge EF, respectively, are applied to the beam.

### Solution Type

Static

### Comparison of Results

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                          | Theory   | Autodesk Nastran | Error (%) |
|------------------------------------------------------|----------|------------------|-----------|
| Translation in X-direction – Axial (in)              | 3.0E-5   | 3.0E-5           | 0.0       |
| Translation in Z-direction – In-Plane Shear (in)     | 1.081E-1 | 1.072E-1         | 0.8       |
| Translation in Y-direction – Out-of-Plane Shear (in) | 4.321E-1 | 4.233E-1         | 2.0       |
| Translation in Y-direction – Twist (in)              | 3.21E-3  | 2.86E-3          | 10.9      |

### References

1. McNeal, R. H. and Harder, R. C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*. North Holland, 1985.

## 2.35 Curved Beam Using Solids

### Problem Description

Figure 1 shows the curved beam model. Static analysis is performed on the beam. The displacement at the free end in the direction of the loads are determined. All dimensions are in inches.

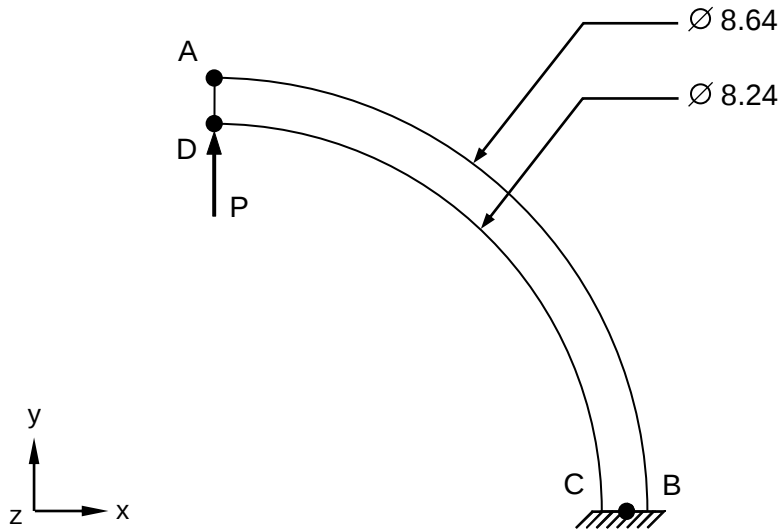


Figure 1. Beam Model

### Autodesk Nastran Analysis Model Filename

- vm2\_35.nas

### Model Data

#### Finite Element Modeling

- 84 nodes, 20 solid elements

#### Units

inch/Kip/second

#### Model Geometry

Inner Radius:  $R_i = 4.12$  in

Outer Radius:  $R_o = 4.32$  in

Width:  $w = 0.1$  in

Height:  $h = 0.2$  in

#### Material Properties

Young's Modulus:  $E = 1.0 \text{ E}+7$  ksi

Poisson's Ratio:  $\nu = 0.25$

**Boundary Conditions**

At clamped end (edge BC) the nodes are restrained in all translations (pinned). This will produce the restraint couples to simulate a clamped condition. A unit load P in the Y-direction is applied at the tip (0.25 Kip/node). Note that the total force is 1 Kip at the tip.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                     | Theory   | Autodesk Nastran | Error (%) |
|---------------------------------|----------|------------------|-----------|
| Translation in Y-direction (in) | 8.854E-2 | 8.816E-2         | 3.8       |

**References**

1. Young, W. C., *Roark's Formulas for Stress and Strain*, 6<sup>th</sup> Edition. New York: McGraw-Hill Co., 1989.

## 2.36 Cantilever Modeled with Variable Thickness Shells and Membranes

### Problem Description

Figure 1 shows the tapered plate model. The thickness varies linearly from 3 inches at the fixed edge to 1 inch at the free end. Static analysis is performed on the plate. The tip displacement in the Z-direction and the tip rotation in the X-rotation due to the uniform moment, together with the tip displacement in the Y-direction due to the tension force are determined. All dimensions are in inches.

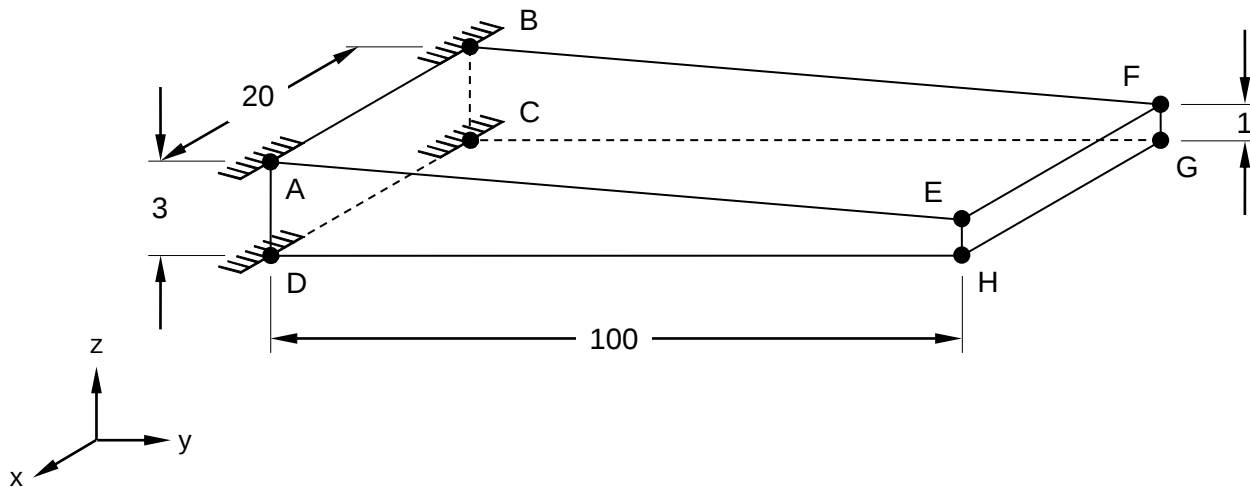


Figure 1. Tapered Plate Model

### Autodesk Nastran Analysis Model Filename

- vm2\_36.nas

### Model Data

#### Finite Element Modeling

- 12 nodes, 5 5-DOF/node quadrilateral elements with varying thickness

#### Units

inch/Kip/second

#### Model Geometry

Length:  $L = 100.0$  in

Width:  $w = 20.0$  in

Height:  $h = 3.0$  in at fixed end, and 1.0 in at the free end

#### Material Properties

Young's Modulus:  $E = 1.0 \text{ E}+3$  ksi

Poisson's Ratio:  $\nu = 0.3$



### Boundary Conditions

At fixed end the nodes are restrained in all translations and rotations. A uniform moment  $M = 3$  in-Kip/in about the X-direction at the free end is applied as 2 moments of 30 in-Kip at each node. A tensile force in the Y-direction  $P = 500$  Kips per node (uniform edge load of 50 Kips/in) is applied at the free end as a second load case.

### Solution Type

Static

### Comparison of Results

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                           | Theory | Autodesk Nastran | Error (%) |
|-------------------------------------------------------|--------|------------------|-----------|
| Translation in Z-direction due to Uniform Moment (in) | 20.0   | 20.1             | 0.5       |
| Rotation in X-direction due to Uniform Moment (rad)   | 0.80   | 0.76             | 5.0       |
| Translation in Y-direction due to Tension Force (in)  | 2.7465 | 2.7225           | 0.9       |

### References

1. Young, W. C., *Roark's Formulas for Stress and Strain*, 6<sup>th</sup> Edition. New York: McGraw-Hill Co., 1989.

## 2.37 Cantilever Modeled with Variable Thickness Solids

### Problem Description

Figure 1 shows the solid model. This is the same problem as problem 2.36, using solids rather than shells. The thickness varies linearly from 3 inches at the fixed edge to 1 inch at the free end. Static analysis is performed on the solid. The tip displacement in the Z-direction due to the uniform moment is determined. All dimensions are in inches.

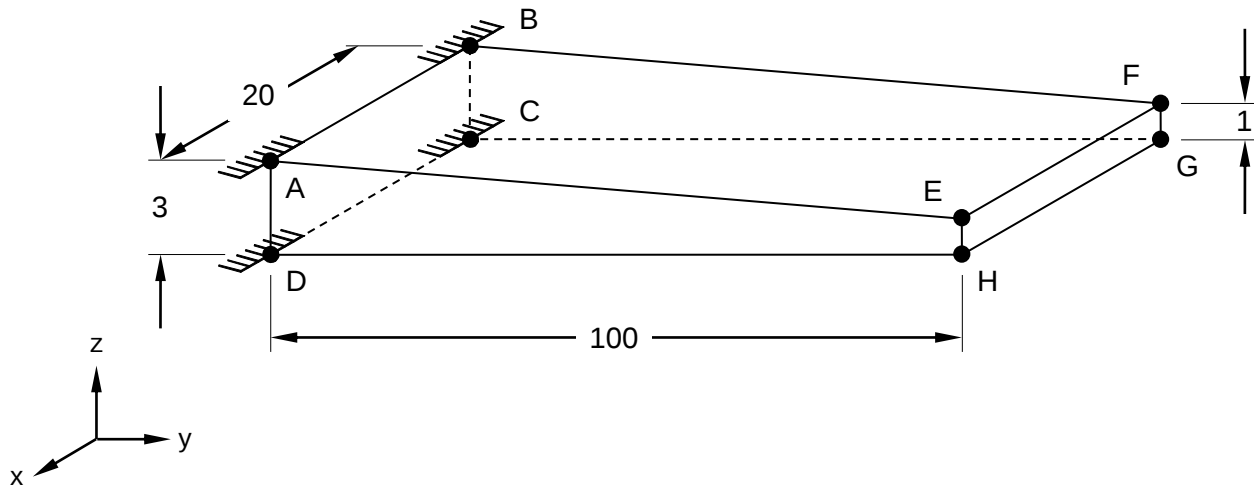


Figure 1. Tapered Solid Model

### Autodesk Nastran Analysis Model Filename

- vm2\_37.nas

### Model Data

#### Finite Element Modeling

- 210 nodes, 80 solid elements with varying thickness

#### Units

inch/Kip/second

#### Model Geometry

Length:  $L = 100.0$  in

Width:  $w = 20.0$  in

Height:  $h = 3.0$  in at fixed end, and  $1.0$  in at the free end

#### Material Properties

Young's Modulus:  $E = 1.0 \text{ E}+3$  ksi

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

At fixed end the nodes are restrained in all translations and rotations. A uniform moment  $M = 3$  in-Kip/in about the X-direction at the free end is applied as moments of 7.5 and 15 in-Kip simulated by a force couple at each external end nodes and internal edge nodes, respectively.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                           | Theory | Autodesk Nastran | Error (%) |
|-------------------------------------------------------|--------|------------------|-----------|
| Translation in Z-direction due to Uniform Moment (in) | 20.0   | 19.7             | 1.5       |

**References**

1. Young, W. C., *Roark's Formulas for Stress and Strain*, 6<sup>th</sup> Edition. New York: McGraw-Hill Co., 1989.

## 2.38 Elongation of a Solid Bar

### Problem Description

Figure 1 shows the tapered solid bar model suspended from the ceiling. The thickness varies linearly from a 2 x 2 inch square at the fixed edge to a 1 x 1 inch square at the free end. Static analysis is performed on the solid. The maximum tip displacement in the Y-direction and the axial stress at mid point (E) are determined. All dimensions are in inches.

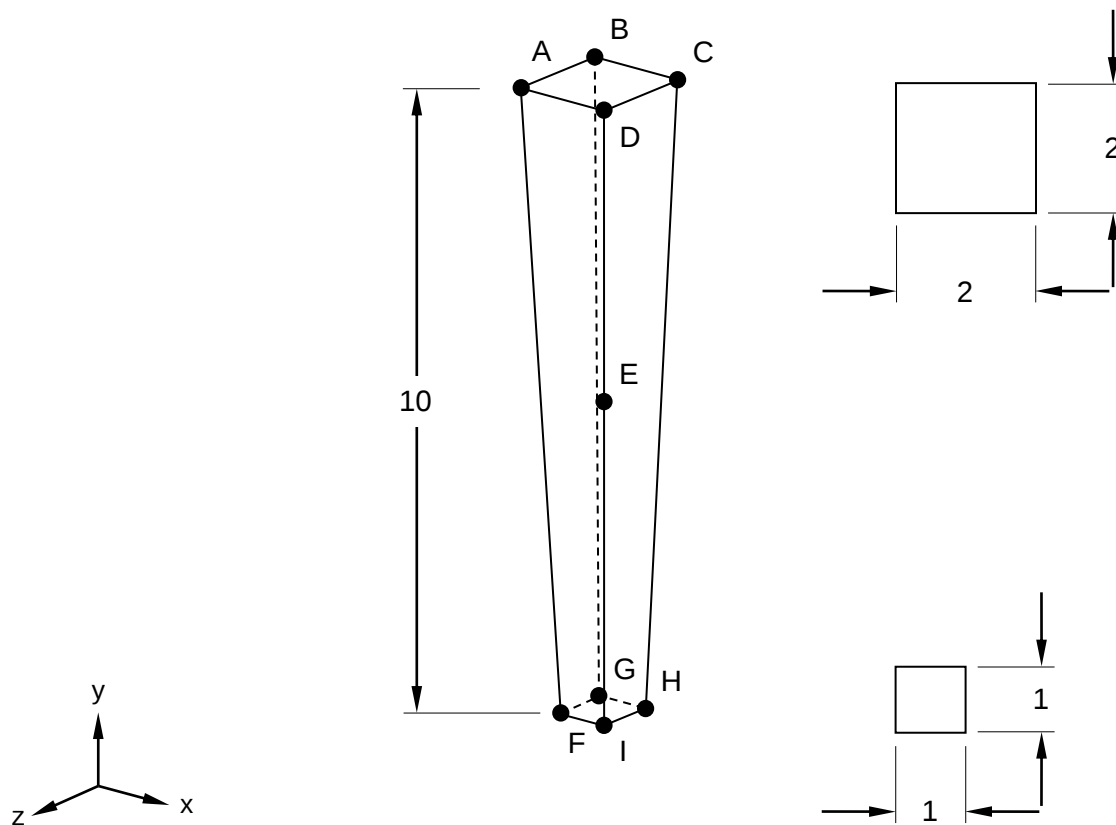


Figure 1. Tapered Solid Bar Model

### Autodesk Nastran Analysis Model Filename

- vm2\_38.nas

### Model Data

#### *Finite Element Modeling*

- 32 nodes, 7 solid elements with varying thickness

#### *Units*

inch/pound/second

**Model Geometry**

Length:  $L = 10.0$  in

Width:  $w = 5.0$  in

Height:  $h = 0.1$  in

**Material Properties**

Young's Modulus:  $E = 10.4 \text{ E}+3$  ksi

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

At fixed end the nodes are restrained in all translations and rotations (points A, B, C and D). A uniform force  $P = 10,000$  lbs in the negative Y-direction ( 2,500 lbs/node) is applied at the free end.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                       | Theory     | Autodesk Nastran | Error (%) |
|-----------------------------------|------------|------------------|-----------|
| Translation in Y-direction (in)   | -4.8077E-3 | -4.757E-3        | 1.1       |
| Axial Stress at Mid Point C (psi) | 4,444      | 4,441            | 0.1       |

**References**

1. Harris, C. O., *Introduction to Stress Analysis*. New York: The Macmillan Co., 1959.

## 2.39 Thin Shell Beam in Pure Bending

### Problem Description

Figure 1 shows the thin flat plate fixed to a wall. Static analysis is performed on the shell beam. The maximum major stress, maximum displacement and strain energy are determined. All dimensions are in inches.

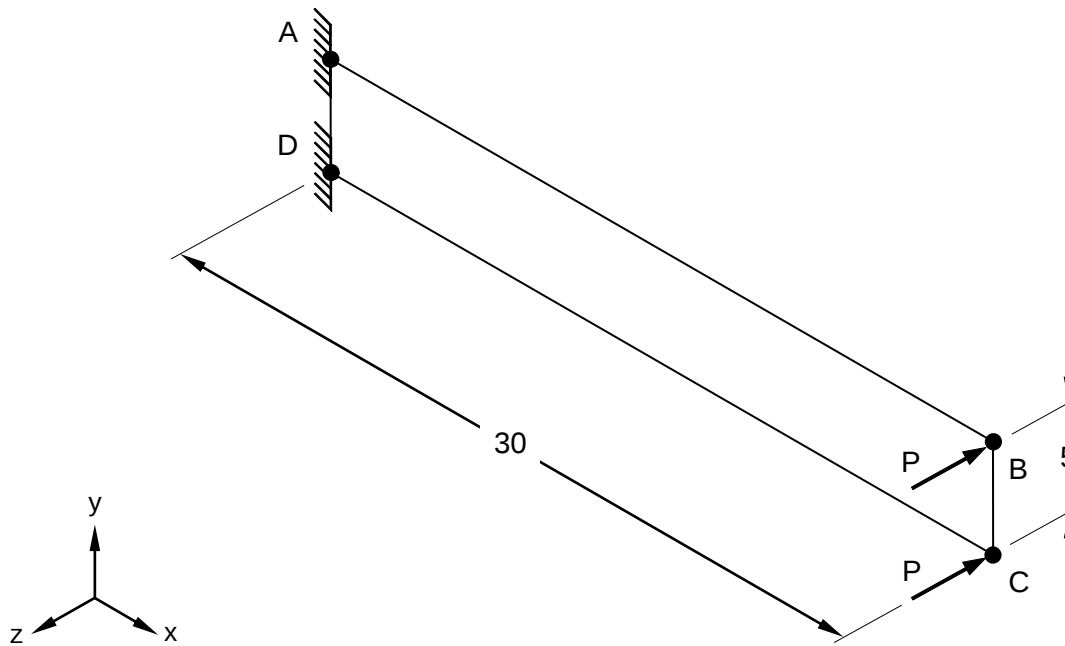


Figure 1. Shell Beam Model

### Autodesk Nastran Analysis Model Filename

- vm2\_39.nas

### Model Data

#### *Finite Element Modeling*

- 14 nodes, 6 5-DOF/node quadrilateral plate elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 30.0$  in

Width :  $w = 0.1$  in

Height:  $h = 5.0$  in

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+3 \text{ ksi}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

At fixed end (edge AD) the nodes are restrained in all translations and rotations. A nodal force  $P = 3 \text{ lbs}$  at each node (edge BD) is applied in the negative Z-direction (total force is 6 lbs).

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                              | Theory | Autodesk Nastran | Error (%) |
|------------------------------------------|--------|------------------|-----------|
| Bottom Major Stress at Node 7 (psi)      | 21,600 | 20,535           | 4.9       |
| Translation in Z-direction at Node 1(in) | 4.320  | 4.255            | 1.5       |
| Total Strain Energy (lb-in)              | 12.96  | 12.77            | 1.5       |

**References**

1. Shigley, J. and Mitchel, L., *Mechanical Engineering Design*, 4<sup>th</sup> Edition. New York: McGraw-Hill Co., 1983.

## 2.40 Static Analysis of Thermal Loading

### Problem Description

Figure 1 shows the pipe line with two right angles subjected to a large temperature change. Static analysis is performed on the pipe model. The support reactions are determined (axial force, shear and moment reaction). All dimensions are in inches.

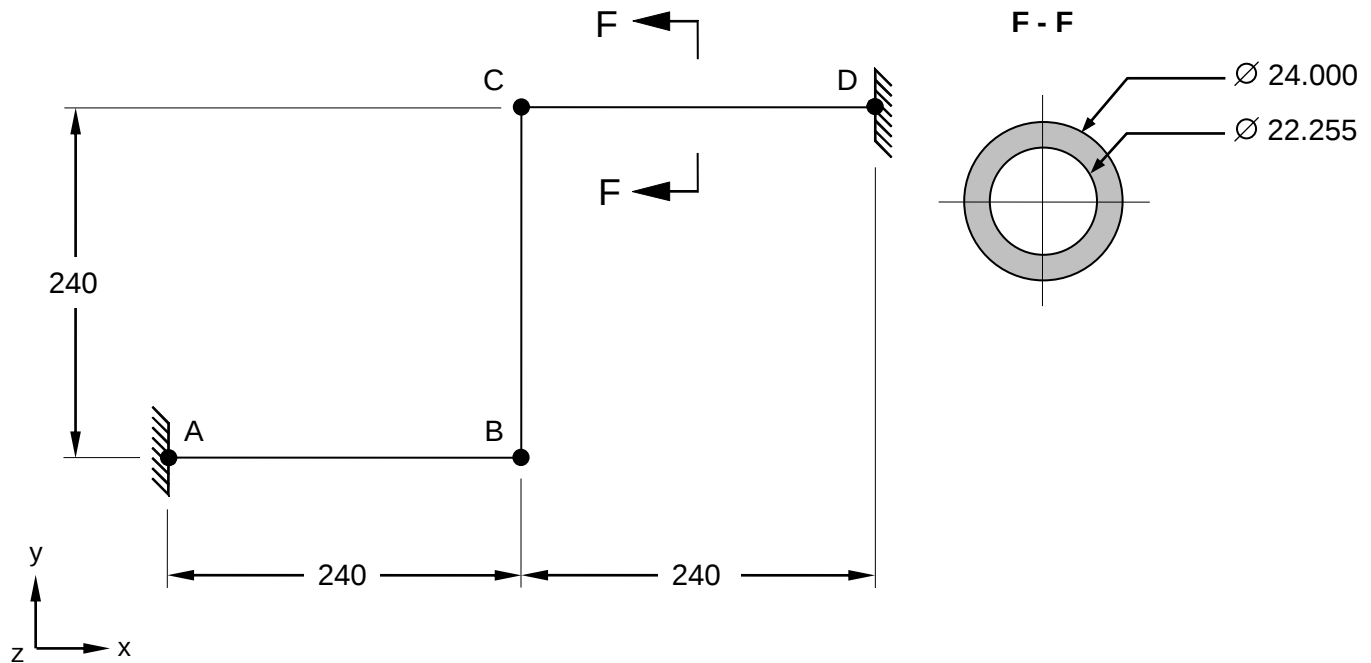


Figure 1. Pipe Model

### Autodesk Nastran Analysis Model Filename

- vm2\_40.nas

### Model Data

#### *Finite Element Modeling*

- 4 nodes, 3 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 240$  in (each pipe)

Outer Radius:  $R_o = 12$  in

Thickness:  $t = 0.8725$  in



### Cross Sectional Properties

Area:  $A_x = 30.5 \text{ in}^2$

Torsional Constant:  $J = 950.0 \text{ in}^4$

Moment of Inertia:  $I = 475.0 \text{ in}^4$

Thermal Expansion Coefficient:  $\alpha = 7.26744 \text{ E-6 in/in/deg F}$

### Material Properties

Young's Modulus:  $E = 26.4 \text{ E+6 psi}$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The model is fully restrained in all translations and rotations at each end of the pipe line (nodes A and D). The other 2 intermediary nodes (B and C) are constrained in the Z-translation, X and Y-rotations. A temperature increase of 430 degrees F to each element is applied. Note that reactions at both supports should be the same.

### Solution Type

Static

### Comparison of Results

Tabular results are given in Table 1.

**Table 1. Results**

| Description             | Theory     | Autodesk Nastran | Error (%) |
|-------------------------|------------|------------------|-----------|
| Axial Force (lb)        | 8,980.47   | 8,949.17         | 0.3       |
| Shear Reaction (lb)     | 7,755.86   | 7,729.19         | 0.4       |
| Moment Reaction (in-lb) | 783,750.00 | 781,105.00       | 0.3       |

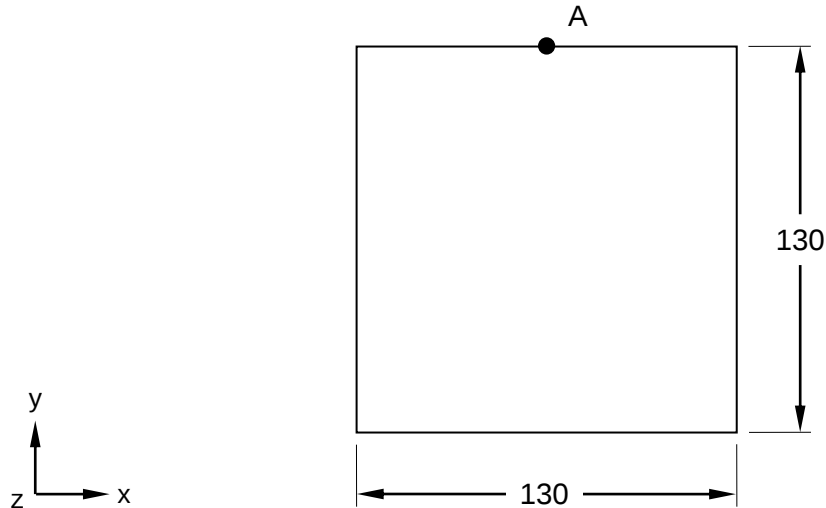
### References

1. Seely, F. B. and Smith, J. O., *Advanced Mechanics of Materials*, 2<sup>nd</sup> Edition. John Wiley and Sons, 1955.

## 2.41 Static Analysis of Pressure Loading

### Problem Description

Figure 1 shows the square plate model. Static analysis is performed on the model with different mesh sizes. The stress at point A is determined for all models. All dimensions are in inches.



**Figure 1. Plate Model**

### Autodesk Nastran Analysis Model Filename

- vm2\_41a4.nas – 5-DOF/node, CQUAD4 elements
- vm2\_41b4.nas – 5-DOF/node, CQUAD4 elements
- vm2\_41c4.nas – 5-DOF/node, CQUAD4 elements

### Model Data

#### *Finite Element Modeling*

- Test vm2\_41a4: 16x16 mesh, 289 nodes, 256 5-DOF/node quadrilateral plate elements
- Test vm2\_41b4: 32x32 mesh, 1089 nodes, 1024 5-DOF/node quadrilateral plate elements
- Test vm2\_41c4: 64x64 mesh, 4225 nodes, 4096 5-DOF/node quadrilateral plate elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 130$  in

Thickness:  $t = 1$  in

### Material Properties

Young's Modulus:  $E = 2.9 \text{ E}+7 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The model is fully restrained in all translations and rotations around the perimeter. A pressure load of 10 psi is applied on the plate.

### Solution Type

Static

### Comparison of Results

Tabular results are given in Table 1.

**Table 1. Results**

| Mesh Size | Element ID | Bench Value  | Autodesk Nastran |           |
|-----------|------------|--------------|------------------|-----------|
|           |            | Stress (psi) | Stress (psi)     | Error (%) |
| 16x16     | 300        | 52,020       | 48,635           | 6.5       |
| 32x32     | 1196       | 52,020       | 50,417           | 3.1       |
| 64x64     | 4780       | 52,020       | 51,178           | 1.6       |

**Note:** The same stress values are obtained with CQUADR elements.

### References

1. Hsu, T. H., *Stress and Strain Data Handbook*. Houston, London, Paris, Tokyo: Gulf Publishing Company, Book Division.

### 3. Linear Statics Verification Using Standard NAFEMS Benchmarks

The purpose of these linear static test cases is to verify the functionality of Autodesk Nastran using standard benchmarks published by NAFEMS (National Agency for Finite Element Methods and Standards, National Engineering Laboratory, Glasgow, U.K.).

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

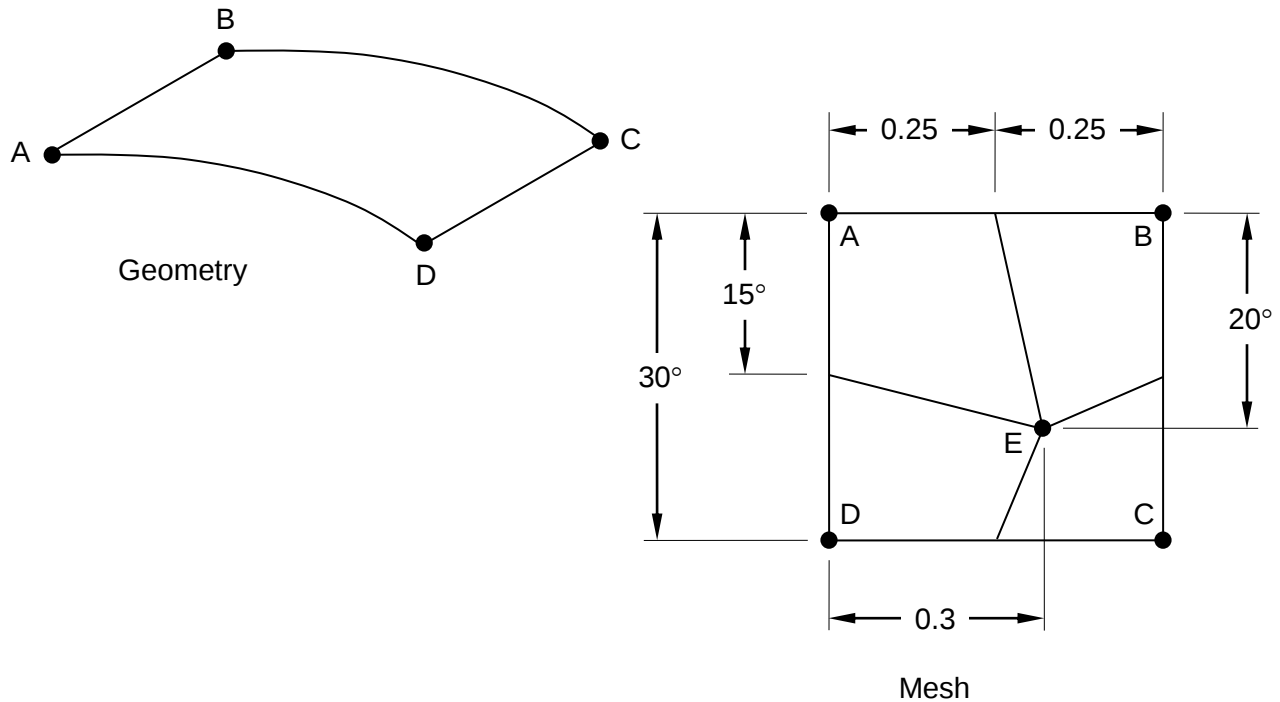
### 3.1 Warped Element Test Cases

The following linear static verification problems using standard NAFEMS Benchmarks are performed using warped plate elements

#### 3.1.1 Cylindrical Shell Patch

##### Problem Description

Figure 1 shows the model of the cylindrical shell patch. A static analysis is performed on the model. The plate top major stress at the center node is determined. All dimensions are in meters.



**Figure 1. Cylindrical Shell Patch**

##### Autodesk Nastran Analysis Model Filename

- vm3\_1\_1a4.nas – Test 1, 5-DOF/node, CQUAD4
- vm3\_1\_1aR.nas – Test 1, 6-DOF/node, CQUADR
- vm3\_1\_1b4.nas – Test 2, 5-DOF/node, CQUAD4
- vm3\_1\_1bR.nas – Test 2, 6-DOF/node, CQUADR

##### Model Data

###### *Finite Element Modeling*

- 9 nodes, 4 5-DOF/node warped quadrilateral plate elements

###### *Units*

meter/Newton/second

### Model Geometry

Length:  $L = 0.5$  m  
 Height:  $h = 0.5$  m  
 Thickness:  $t = 0.01$  m

### Material Properties

Young's Modulus:  $E = 210.0 \text{ E}+3$  MPa  
 Poissons ratio:  $\nu = 0.3$

### Boundary Conditions

Loads Test 1 (vm3\_1\_1a4 and vm3\_1\_1aR): Edge DC is loaded with nodal moments of -125 N-m at point D, -250 N-m at bottom center node, and -125 N-m at point C about the Z-axis.

Loads Test 2 (vm3\_1\_1b4 and vm3\_1\_1bR): Nodal forces of 75,000 N are applied at points D and C, and a force of 150,000 N is applied to the bottom edge middle node in the direction away from and tangential to the element. An elemental pressure load is applied to all elements of 600,000 Pa in the Y-direction.

Constraints Test 1 & 2: The nodes along edge AB are fully constrained in all translations and rotations. The nodes along edges AD and BC are constrained in the Z-translation and X and Y-rotations.

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Test 1 results**

| Description                       | Element | NAFEMS | Autodesk Nastran | Error (%) |
|-----------------------------------|---------|--------|------------------|-----------|
| Plate Top Major Stress at Point E | CQUAD4  | 60.0   | 47.7             | 20.5      |
|                                   | CQUADR  | 60.0   | 50.3             | 16.2      |

**Table 2. Test 2 results**

| Description                       | Element | NAFEMS | Autodesk Nastran | Error (%) |
|-----------------------------------|---------|--------|------------------|-----------|
| Plate Top Major Stress at Point E | CQUAD4  | 60.0   | 61.58            | 2.6       |
|                                   | CQUADR  | 60.0   | 62.4             | 4.0       |

### References

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE2.
2. Davies, G. A. O., Fenner, R. T., and Lewis, R. W., *Background to Benchmarks*. Glasgow: NAFMES, 1993.

### 3.1.2 Hemisphere-Point Loads

#### Problem Description

Figure 1 shows the model of the hemisphere with an acting point load at point A. Static analysis is performed on the model. The T1 translation at point A is determined. All dimensions are in meters.

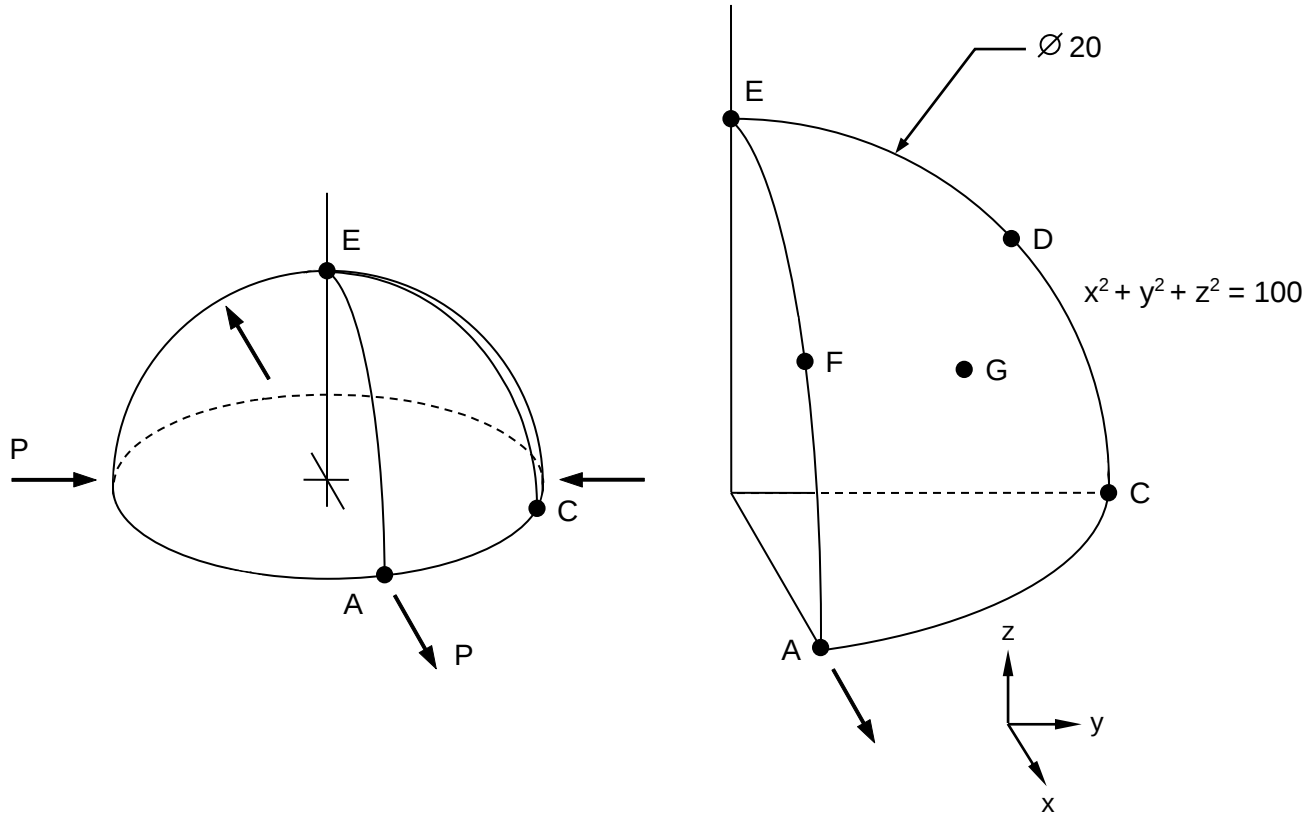


Figure 1. Hemisphere with Point Load

#### Autodesk Nastran Analysis Model File

- vm3\_1\_2b4.nas – 5-DOF/node, CQUAD4
- vm3\_1\_2bR.nas – 6-DOF/node, CQUADR

#### Model Data

##### Finite Element Modeling

- 61 nodes, 48 warped quadrilateral plate elements. Point G is at  $X = Y = Z = \frac{10}{\sqrt{3}}$

#### Units

meter/Newton/second

**Model Geometry**

Thickness:  $t = 0.04$  m

Radius:  $R = 10$  m

**Material Properties**

Young's Modulus:  $E = 68.25 \text{ E}+3$  MPa

Poissons Ratio:  $\nu = 0.3$

**Boundary Conditions**

Point E is fully constrained in all translations and rotations. Symmetry boundary conditions are applied to edges AE and CE. An outward radial load  $P = 2$  kN is applied to point A and an inward radial load  $P = 2$  kN is applied to point C.

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                   | Element | NAFEMS | Autodesk Nastran | Error (%) |
|-------------------------------|---------|--------|------------------|-----------|
| T1 Translation at Point A (m) | CQUAD4  | 0.185  | 0.179            | 3.2       |
|                               | CQUADR  | 0.185  | 0.170            | 8.1       |

**References**

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE3.
2. Davies, G. A. O., Fenner, R. T., and Lewis, R. W., *Background to Benchmarks*. Glasgow: NAFMES, 1993.



3.2 Laminate Plate Element Test Cases

The following linear static verification problems using standard NAFEMS Benchmarks are performed using laminate plate elements.

3.2.1 Laminated Strip

Problem Description

Figure 1 shows the model of the laminated strip. A static analysis is performed on the model. The interlaminar shear stress at D and the Z-deflection at E are determined. All dimensions are in millimeters.

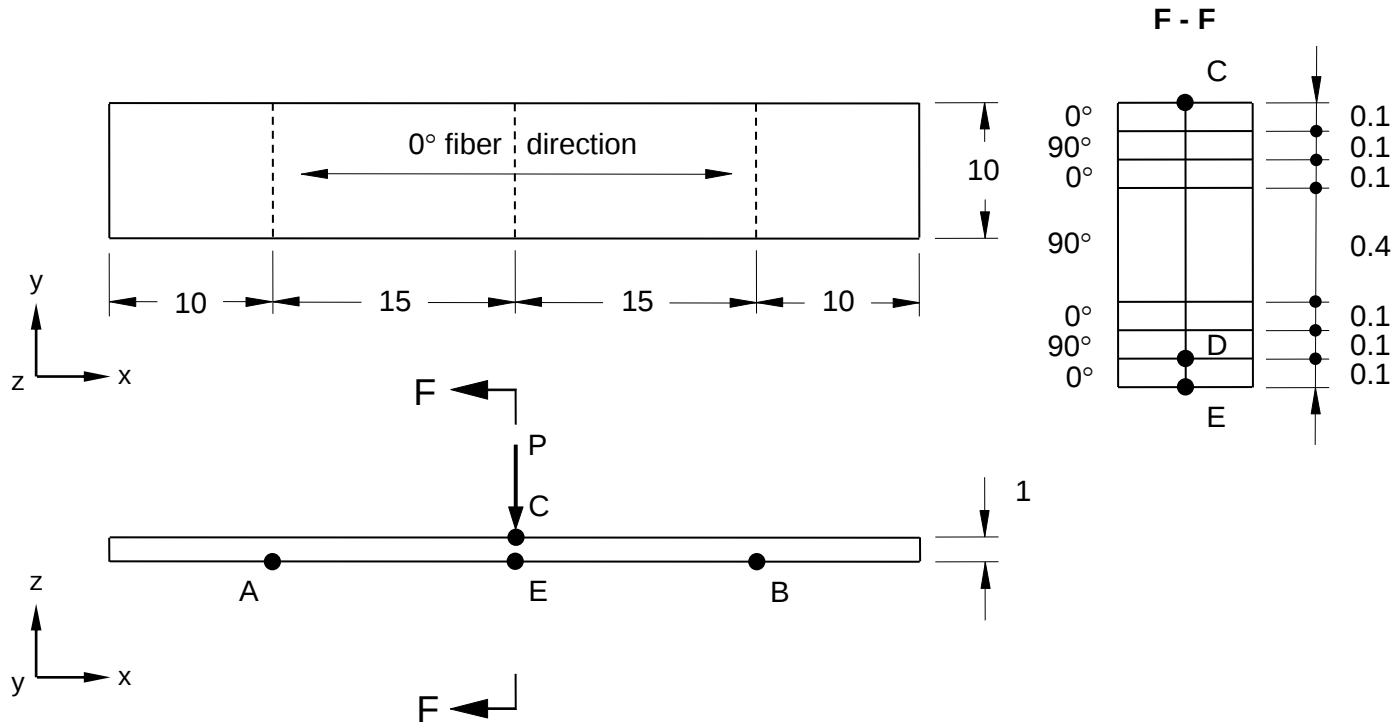


Figure 1. Laminated Strip

Autodesk Nastran Analysis Model Filename

- vm3\_2\_1.nas

Model Data

Finite Element Modeling

- 33 nodes, 20 5-DOF/node laminate plate elements (2x10 elements, on the smaller and larger sides of the plate, respectively).

Units

meter/Newton/second

**Model Geometry**

Length:  $L = 0.050$  m

Width:  $w = 0.010$  m

Thickness:  $t = 0.001$  mm

**Material Properties**

Young's Modulus:  $E_1 = 1.0 \text{ E}+5$  MPa

Young's Modulus:  $E_2 = 5.0 \text{ E}+3$  MPa

Poisons Ratio:  $\nu_{12} = 0.4$

Poisons Ratio:  $\nu_{23} = 0.3$

Shear Modulus of Elasticity:  $G_{12} = 3.0 \text{ E}+3$  MPa

Shear Modulus of Elasticity:  $G_{13} = G_{23} = 2.0 \text{ E}+3$  MPa

**Boundary Conditions**

The model is simply supported at points A and B. A distributed load  $P = 10\text{N/mm}$  is applied at point C ( $X=25$ ,  $Z=1$ ).

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                          | NAFEMS | Autodesk Nastran | Error (%) |
|--------------------------------------|--------|------------------|-----------|
| Interlaminar Shear Stress at D (MPa) | -4.1   | -4.1             | 0.0       |
| Deflection in Z-direction at E (mm)  | -1.06  | -1.06            | 0.0       |

**References**

1. NAFEMS Report R0031, *Laminated Strip*. Test No. R0031/1, date issued 17/12/98/1.

### 3.2.2 Wrapped Thick Cylinder

#### Problem Description

Figure 1 shows the geometry of the cylinder. A static analysis is performed on the model (an 8<sup>th</sup> part of the cylinder). The hoop stresses in the inner and outer cylinders are determined. All dimensions are in millimeters.

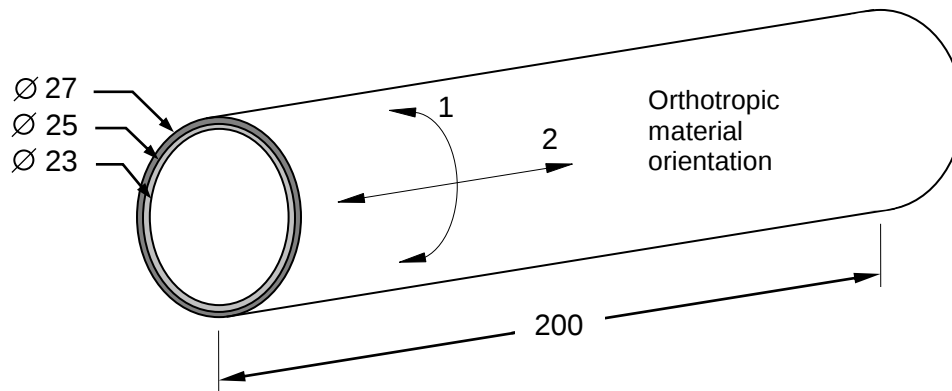


Figure 1. Wrapped Thick Cylinder

#### Autodesk Nastran Analysis Model Filename

- vm3\_2\_2.nas

#### Model Data

##### *Finite Element Modeling*

- 33 nodes, 100 5-DOF/node laminate plate elements (10x10 elements, on an 8<sup>th</sup> part of the cylinder model, using symmetry)

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L = 0.200$  m

Radius:  $R_1 = 0.023$  m

Radius:  $R_2 = 0.025$  m

Radius:  $R_3 = 0.027$  m

##### *Material Properties*

Inner Cylinder Isotropic

Young's Modulus:  $E = 2.1 \text{ E}+5$  MPa

Poisons Ratio:  $\nu = 0.3$

Outer Cylinder Circumferentially Wound

Young's Modulus:  $E_1 = 1.3 \text{ E}+5 \text{ MPa}$

Young's Modulus:  $E_2 = 5.0 \text{ E}+3 \text{ MPa}$

Poisons Ratio:  $\nu = 0.25$

Shear Modulus of Elasticity:  $G_{12} = 1.0 \text{ E}+4 \text{ MPa}$

Shear Modulus of Elasticity:  $G_{13} = G_{23} = 5.0 \text{ E}+3 \text{ MPa}$

### Boundary Conditions

Symmetry boundary conditions are applied to the 8<sup>th</sup> model cylinder.

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                   | NAFEMS  | MSC.Nastran | Autodesk Nastran | Error (%) |
|-------------------------------|---------|-------------|------------------|-----------|
| Hoop Stress in Inner Cylinder | 1,497.5 | 1,518.1     | 1,518.2          | 1.4       |
| Hoop Stress in Outer Cylinder | 816.9   | 975.2       | 975.2            | 19.4      |

### References

1. NAFEMS Report R0031, *Wrapped Thick Cylinder*. Test No. R0031/2, date issued 17/12/98/1.

### 3.3 Shell Element Test Cases

The following linear static verification problems using standard NAFEMS Benchmarks are performed using shell plate elements.

#### 3.3.1 Elliptic Membrane

##### Problem Description

Figure 1 shows the membrane model. Static analysis is performed on the elliptical shell model. The normal stress at point C is determined. All dimensions are in meters.

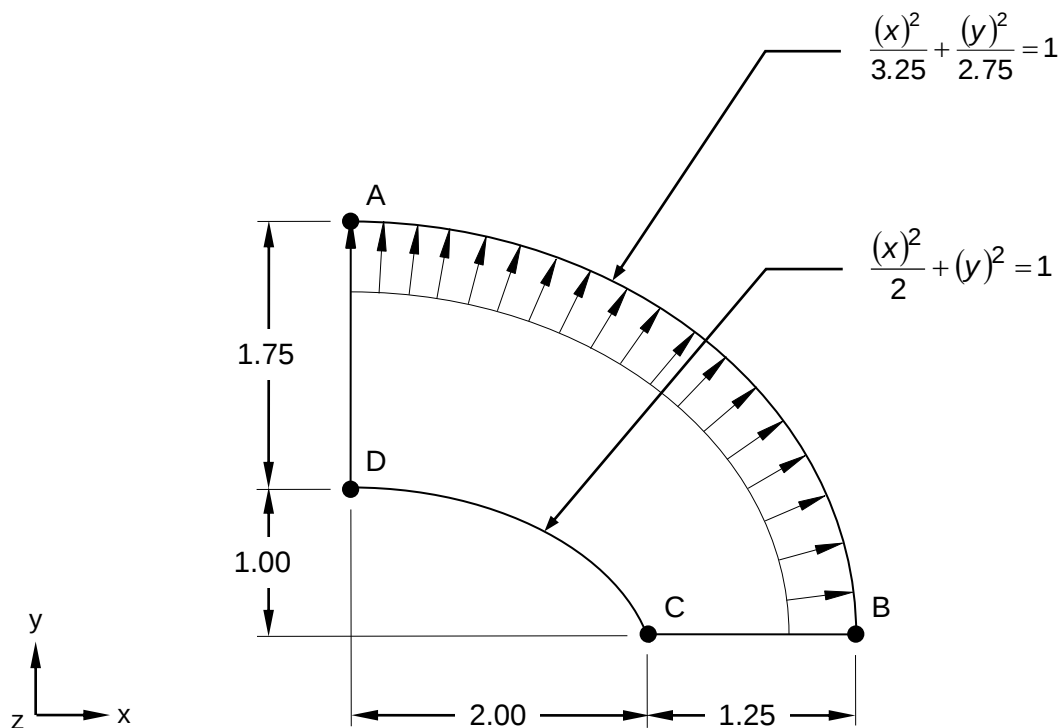


Figure 1. Elliptic Membrane

##### Autodesk Nastran Analysis Model Filename

- vm3\_3\_1.nas

##### Model Data

##### Finite Element Modeling

- A  $\frac{1}{4}$  model (90 degrees sector of two ellipses) is created using 231 nodes, 200 5-DOF/node quadrilateral elements and 20 bar elements on outer edge AB (used to transfer load to the outer edge nodes in the proper direction).

**Units**

meter/Newton/second

**Model Geometry**

Length:  $L_1 = 1.75$  m and  $L_2 = 1.25$  m

Thickness:  $t = 0.1$  m

**Material Properties**

Young's Modulus:  $E = 210.0 \text{ E}+3$  MPa

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All nodes along edge AD are restrained in the X and Z-translations, and X and Y-rotations. All nodes along edge BC are restrained in the in the Y and Z-translations, and X and Y-rotations. A uniform outward pressure  $P = 10$  MPa at outer edge AB is applied to the model.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                      | Theory | Autodesk Nastran | Error (%) |
|--------------------------------------------------|--------|------------------|-----------|
| Tangential (Normal) Edge Stress at Point C (MPa) | 92.7   | 91.2             | 1.6       |

**References**

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE1.

### 3.3.2 Z-Section Cantilever

#### Problem Description

Figure 1 shows the Z-Section cantilever model. Static analysis is performed on the shell model. The plate minimal principal stress at point A is determined. All dimensions are in meters.

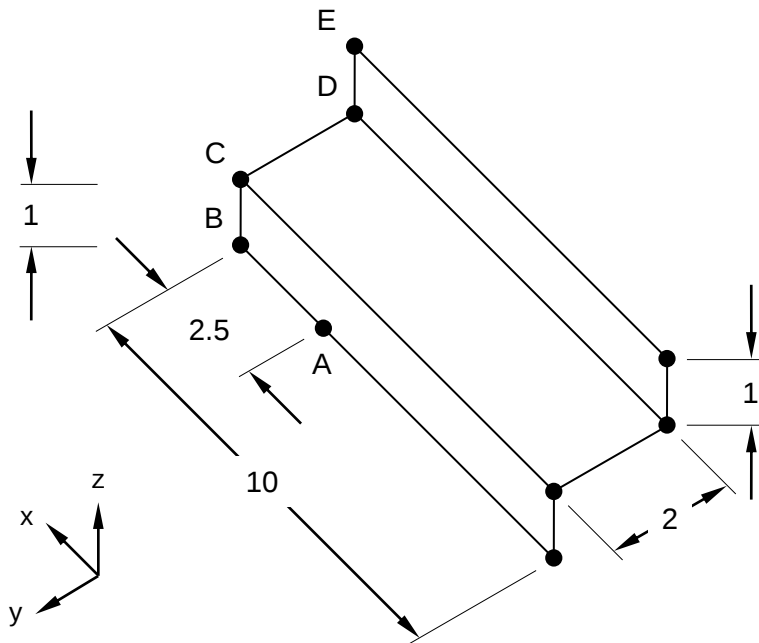


Figure 1. Z-Section Cantilever

#### Autodesk Nastran Analysis Model Filename

- vm3\_3\_2.nas

#### Model Data

##### *Finite Element Modeling*

- 36 nodes, 24 5-DOF/node quadrilateral elements

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L = 10.0$  m

Width:  $w = 2.0$  m

Height:  $h = 1.0$  m

Thickness:  $t = 0.1$  m

**Material Properties**

Young's Modulus:  $E = 210.0 \text{ E}+3 \text{ MPa}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

The nodes at B, C, D, and E are restrained in all translations and rotations. A torque of 1.20 MN-m is applied at the free end by way of two uniformly distributed edge shears  $P = 0.60 \text{ MN}$  at each flange.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                 | Theory | Autodesk Nastran | Error (%) |
|---------------------------------------------|--------|------------------|-----------|
| Shell Min Principal Stress at Point A (MPa) | -108   | -108             | 0.0       |

**References**

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE5.



### 3.3.3 Skew Plate Under Normal Pressure

#### Problem Description

Figure 1 shows the skew plate model. Static analysis is performed on the shell model. The plate bottom major stress at center of plate is determined. All dimensions are in meters.

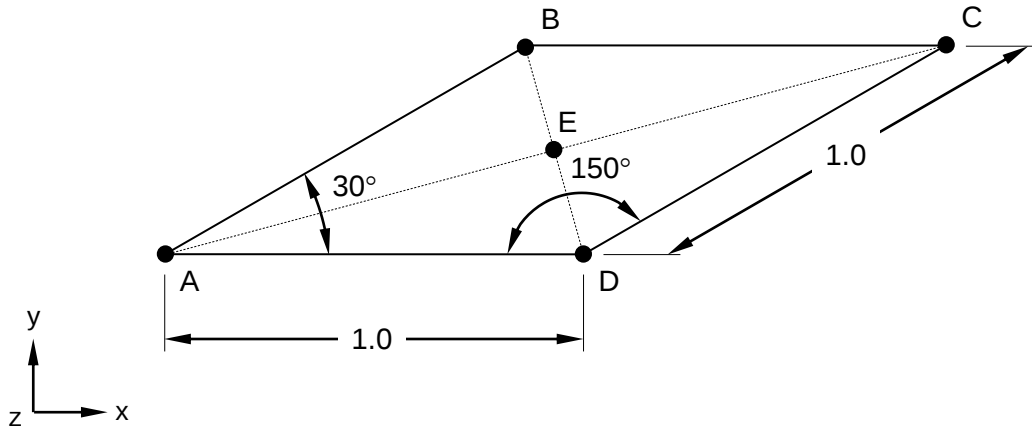


Figure 1. Skew Plate

#### Autodesk Nastran Analysis Model Filename

- vm3\_3\_3.nas

#### Model Data

##### *Finite Element Modeling*

- 81 nodes, 64 6-DOF/node quadrilateral elements

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L = 1$  m

Thickness:  $t = 0.01$  m

Internal Skew Angles:  $\theta = 30^\circ$  and  $150^\circ$

##### *Material Properties*

Young's Modulus:  $E = 210.0 \text{ E}+3$  MPa

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All nodes along all edges are restrained in the Z-translation. Point A has X and Y-translations restrained, while point D has Y-translation restrained. A uniform pressure  $P = 700$  Pa is applied in the negative Z-direction.

**Solution Type**

Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                  | Theory | Autodesk Nastran | Error (%) |
|----------------------------------------------|--------|------------------|-----------|
| Bottom Major Stress at Center of Plate (MPa) | 0.802  | 0.769            | 4.1       |

**References**

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE6.

#### 4. Normal Modes/Eigenvalue Verification Using Theoretical Solutions

The purpose of these normal mode dynamic test cases is to verify the functionality of Autodesk Nastran using theoretical solutions of well-known engineering normal mode dynamic problems. The test cases are basic in form and most of them have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

## 4.1 Two Degree of Freedom Undamped Free Vibration – Principle Modes

### Problem Description

Figure 1 shows the model of the two degree of freedom system. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the system are determined.

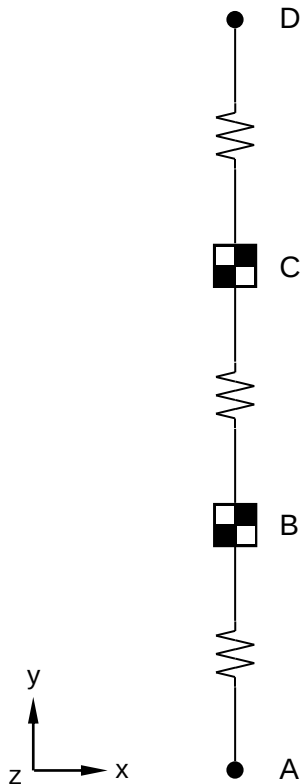


Figure 1. Two Degree of Freedom System

### Autodesk Nastran Analysis Model Filenames

- vm4\_1d.nas – Diagonal mass formulation
- vm4\_1c.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- Four nodes are created on the Y-axis (A, B, C, D). Three DOF springs are created with stiffness of 1 N/m and a stiffness reference coordinate system being uniaxial. Mass elements with a mass of 1 kg are created. DOF springs and mass elements are used.

#### *Units*

meter/Newton/second

### Physical Properties

Mass:  $m = 1$  kg

Spring Constant:  $k = 1$  N/m

### Boundary Conditions

The end nodes (A and D) are constrained in all DOF. The other nodes (B and C) are constrained in all DOF except the Y-translation.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|             | Theory                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 0.1592                 | 0.1592                                     | 0.0       | 0.1592                                    | 0.0       |
| 2           | 0.2757                 | 0.2757                                     | 0.0       | 0.2757                                    | 0.0       |

### References

1. Tse, F., Morse, I., and Hinkle, R., *Mechanical Vibrations*, 2<sup>nd</sup> Edition. Boston: Allyn and Bacon, Inc., 1978. pp. 145-149.

## 4.2 Three Degree of Freedom Torsional System

### Problem Description

Figure 1 shows the torsional system. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the system are determined.

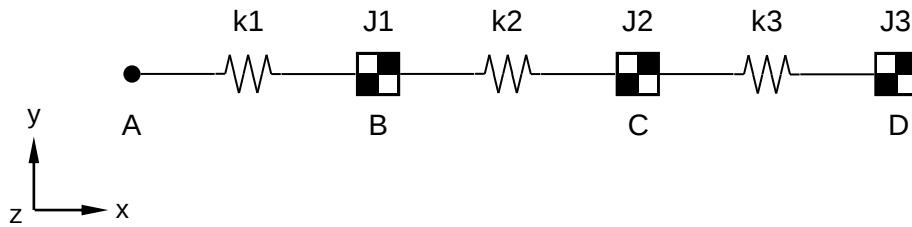


Figure 1. Three Degree of Freedom Torsional System

### Autodesk Nastran Analysis Model Filenames

- vm4\_2d.nas – Diagonal mass formulation
- vm4\_2c.nas – Coupled mass formulation

### Model Data

#### Finite Element Modeling

- Four nodes are created on the X-axis (A, B, C, D). Three DOF springs are created with stiffness of 1 N/m and a stiffness reference coordinate system being uniaxial. Three mass elements are created with a mass coordinate system = 1 and mass inertia system of: 0.1, 0.0, 0.0, 0.0, 0.0, 0.0. DOF springs and mass elements are used.

#### Units

meter/Newton/second

#### Physical Properties

Mass:  $J = J1 = J2 = J3 = 0.1$  (mass)

Spring Constant:  $k = k1 = k2 = k3 = 1$  N-m (stiffness)

#### Boundary Conditions

One end node (node A), is constrained in all translations and rotations. All other nodes are constrained in all translations and rotations except the X-rotation.

#### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

## Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|                | Theory                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1              | 0.2240                       | 0.2240                                        | 0.0       | 0.2240                                       | 0.0       |
| 2              | 0.6276                       | 0.6276                                        | 0.0       | 0.6276                                       | 0.0       |
| 3              | 0.9069                       | 0.9069                                        | 0.0       | 0.9069                                       | 0.0       |

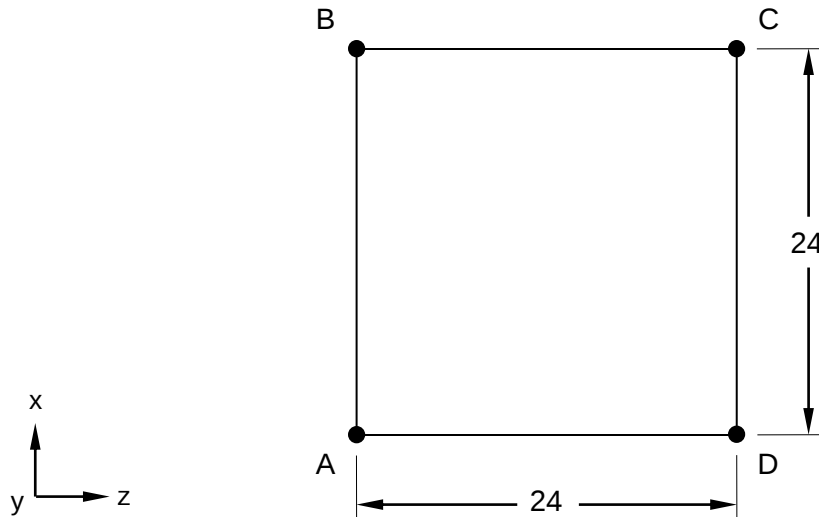
## References

1. Tse, F., Morse, I., and Hinkle, R., *Mechanical Vibrations*, 2<sup>nd</sup> Edition. Boston: Allyn and Bacon, Inc., 1978. pp. 153-155.

### 4.3 Cantilever Plate Eigenvalue Problem

#### Problem Description

Figure 1 shows the cantilever plate. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The first two modes of vibration of the plate are determined. All dimensions are in inches.



**Figure 1. Cantilever Plate**

#### Autodesk Nastran Analysis Model Filenames

- vm4\_3d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm4\_3c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm4\_3dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation
- vm4\_3cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation

#### Model Data

##### *Finite Element Modeling*

- 400 nodes, 361 quadrilateral plate elements

##### *Units*

inch/pound/second

##### *Model Geometry*

*Length:*  $L = 24$  in

*Thickness:*  $t = 1$  in



### Material Properties

Young's Modulus:  $E = 29.5 \text{ E}+3 \text{ ksi}$

Poisson's Ratio:  $\nu = 0.2$

Mass Density:  $\rho = 0.28356 \text{ lbs/in}^3$

### Boundary Conditions

All the nodes on edge AB are restrained in all translations and rotations. The load is mass distribution based on density and given geometry (as nodal forces in the X, Y and Z-directions).

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | Theory                | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|-----------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (s) | Natural Frequency (s)                      | Error (%) | Natural Frequency (s)                     | Error (%) |
| 1           | CQUAD4  | 0.01790               | 0.01757                                    | 1.8       | 0.01755                                   | 2.0       |
|             | CQUADR  | 0.01790               | 0.01757                                    | 1.8       | 0.01755                                   | 2.0       |
| 2           | CQUAD4  | 0.00732               | 0.00696                                    | 4.9       | 0.00695                                   | 5.1       |
|             | CQUADR  | 0.00732               | 0.00696                                    | 4.9       | 0.00695                                   | 5.1       |

### References

1. Harris, C. M., and Crede, C. E., *Shock and Vibration Handbook*. New York: McGraw-Hill, Inc., 1976.

## 4.4 Bathe and Wilson Frame Eigenvalue Problem

### Problem Description

Figure 1 shows the frame model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The first three eigenvalues are determined. All dimensions are in feet.

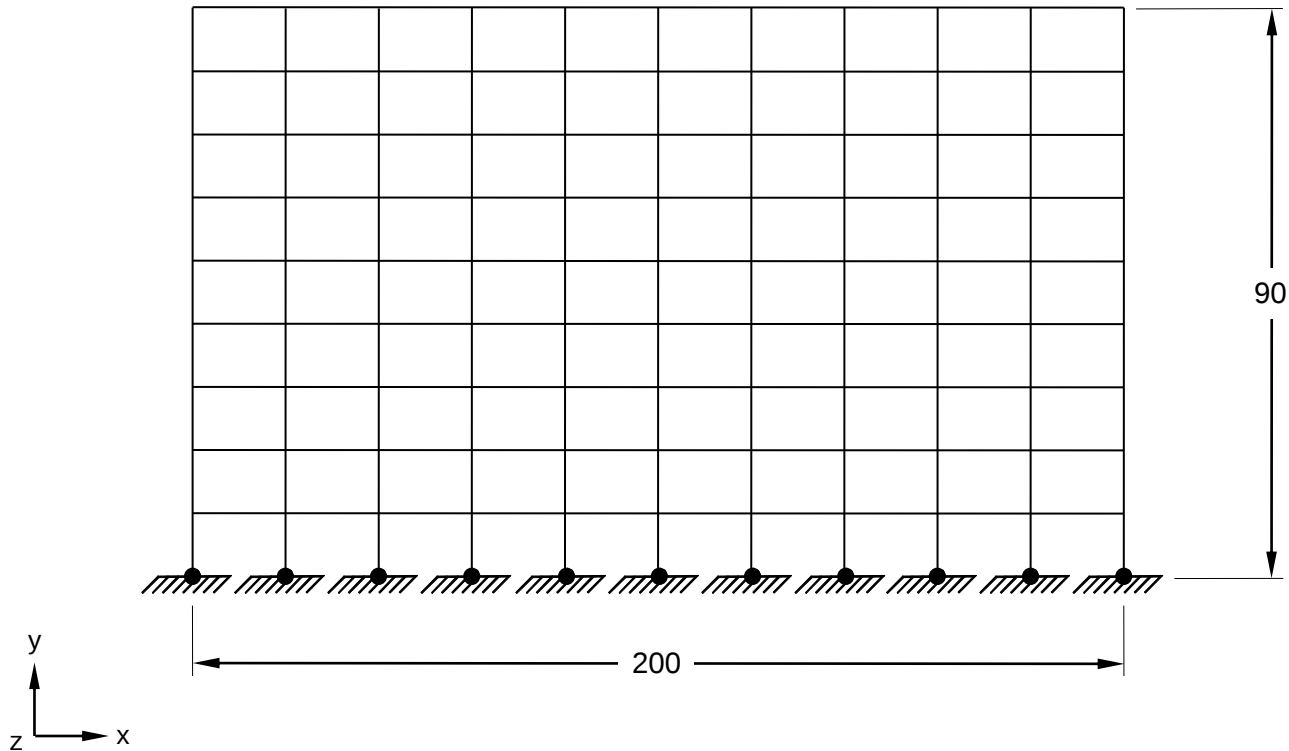


Figure 1. Bathe and Wilson Frame

### Autodesk Nastran Analysis Model Filenames

- vm4\_4d.nas – Diagonal mass formulation
- vm4\_4c.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- 110 nodes, 189 beam elements

#### *Units*

feet/Kip/second

#### *Model Geometry*

Length:  $L = 200$  feet (10 @ 20)

Height:  $h = 90$  feet (9 @ 10)

### Cross Sectional Properties

Area:  $A = 3 \text{ ft}^2$

Moment of Inertia:  $I_z = 1 \text{ ft}^4$  (for each beam and column)

### Material Properties

Young's Modulus:  $E = 4.32 \text{ E}+5 \text{ k/ft}^2$

Poisson's Ratio:  $\nu = 0.3$

Mass per Unit Length:  $m = 3 \text{ kip-sec}^2/\text{ft /ft}$

### Boundary Conditions

The supports are restrained in all translations and rotations. All the other nodes are restrained in the Z-translation, X and Y-rotations. The load is mass distribution based on density and given geometry.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|             | Theory             | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|--------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | Eigenvalue (rad/s) | Eigenvalue (rad/s)                         | Error (%) | Eigenvalue (rad/s)                        | Error (%) |
| 1           | 0.589541           | 0.588858                                   | 0.1       | 0.589827                                  | 0.0       |
| 2           | 5.52695            | 5.47130                                    | 1.0       | 5.55036                                   | 0.4       |
| 3           | 16.5878            | 16.1568                                    | 2.6       | 16.7757                                   | 1.1       |

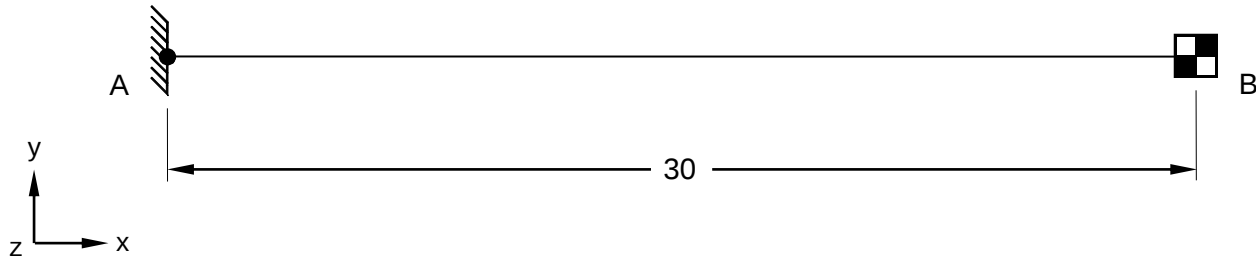
### References

1. Bathe, K. J., and Wilson, E. L., *Large Eigenvalue Problems in Dynamic Analysis*. Journal of the Engineering Mechanics Division, ASCE, Vol. 98, No. EM6, Paper 9433, 1972.

## 4.5 Natural Frequency of a Cantilevered Mass

### Problem Description

Figure 1 shows the cantilevered mass model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequency of vibration of the mass attached to the end of the light cantilever beam is determined. All dimensions are in inches.



**Figure 1. Cantilevered Mass Model**

### Autodesk Nastran Analysis Model Filenames

- vm4\_5d.nas – Diagonal mass formulation
- vm4\_5c.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- 2 nodes, 1 beam element, and 1 mass element

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 30$  in

#### *Cross Sectional Properties*

Moment of Inertia:  $I = 1.33333 \text{ in}^4$

#### *Material Properties*

Young's Modulus:  $E = 30.0 \text{ E}+3 \text{ ksi}$

Poisson's Ratio:  $\nu = 0.3$

Mass:  $m = 0.1 \text{ lb-sec}^2/\text{in}$

**Boundary Conditions**

The beam is fixed at point A (all translations and rotations are restrained). The other end of the beam (point B) is restrained in the Z-translation and X-rotation. The load is a lumped mass at free end of the beam.

**Solution Type**

Normal Modes/Eigenvalue – Subspace iterative method

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

|                | Theory                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1              | 33.553                       | 33.553                                        | 0.0       | 33.553                                       | 0.0       |

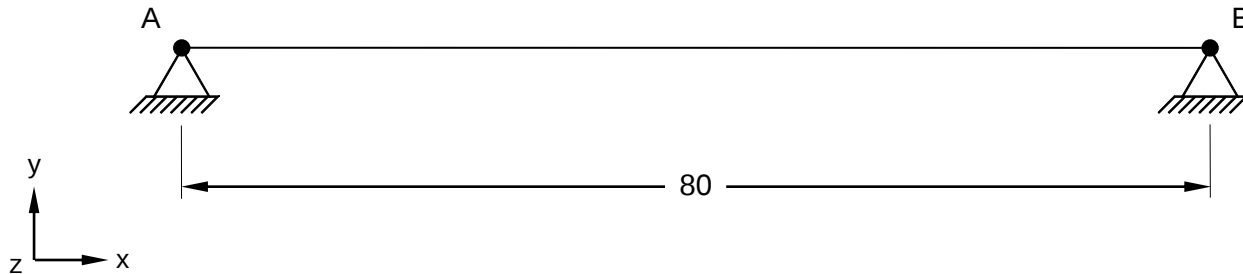
**References**

1. Thompson, W. T., *Vibration Theory and Applications*, 2<sup>nd</sup> Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.

## 4.6 Fundamental Frequency of a Simply Supported Beam

### Problem Description

Figure 1 shows the simply supported beam model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The fundamental frequency of the simply supported beam of uniform cross-section is determined. All dimensions are in inches.



**Figure 1. Simply Supported Beam**

### Autodesk Nastran Analysis Model Filenames

- vm4\_6d.nas – Diagonal mass formulation
- vm4\_6c.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- 5 nodes, 4 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 80$  in

Height:  $h = 2$  in

#### *Cross Sectional Properties*

Area:  $A = 4$  in<sup>2</sup>

Moment of Inertia:  $I = 1.33333$  in<sup>4</sup>

Square Cross Section = (2 in x 2 in)

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+3 ksi

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The beam is pinned at both ends (all translations are restrained). The load is a uniform mass along the beam  $w = 1.124 \text{ lb/in}$  ( $m = 1.124/386.4 = 0.002909 \text{ lb-sec}^2/\text{in}^2$ ).

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|             | Theory                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 28.766                 | 28.520                                     | 0.9       | 28.782                                    | 0.1       |

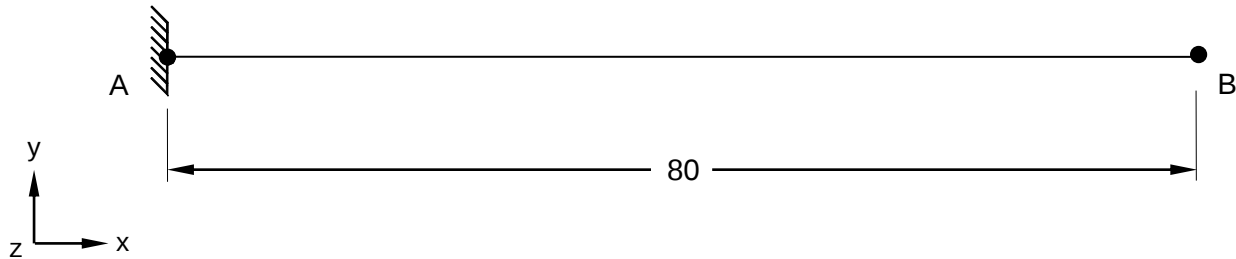
### References

1. Thompson, W. T., *Vibration Theory and Applications*, 2<sup>nd</sup> Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.

## 4.7 Natural Frequencies of a Cantilever Beam

### Problem Description

Figure 1 shows the cantilever beam model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The first three natural frequencies of the cantilever beam of uniform cross-section are determined. All dimensions are in inches.



**Figure 1. Cantilever Beam**

### Autodesk Nastran Analysis Model Filenames

- vm4\_7d.nas – Diagonal mass formulation
- vm4\_7c.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- 21 nodes, 20 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 80$  in

Height:  $h = 2$  in

#### *Cross Sectional Properties*

Area:  $A = 4$  in<sup>2</sup>

Moment of Inertia:  $I = 1.33333$  in<sup>4</sup>

Square Cross Section = (2 in x 2 in)

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+3 ksi

Poisson's Ratio:  $\nu = 0.3$



### Boundary Conditions

The beam is fixed at end A (all translations and rotations are restrained). All other nodes are restrained in the Y-translation, X and Z-rotations. The load is a uniform mass along the beam  $w = 1.124 \text{ lb/in}$  ( $m = 1.124/386.4 = 0.002909 \text{ lb-sec}^2/\text{in}^2$ ).

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|             | Theory                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 10.247                 | 10.239                                     | 0.1       | 10.252                                    | 0.0       |
| 2           | 64.221                 | 63.907                                     | 0.5       | 64.202                                    | 0.0       |
| 3           | 179.82                 | 178.12                                     | 0.9       | 179.56                                    | 0.1       |

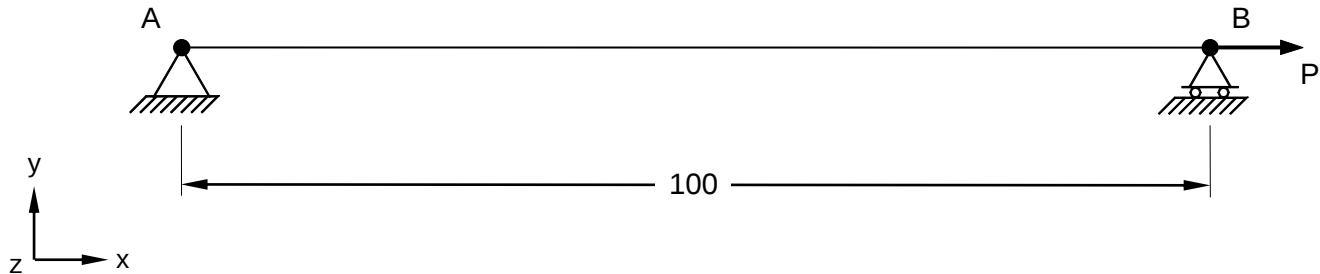
### References

1. Thompson, W. T., *Vibration Theory and Applications*, 2<sup>nd</sup> Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.

## 4.8 Vibration of a String Under Tension

### Problem Description

Figure 1 shows the string (beam) model. A linear prestress modal analysis is performed using the subspace iterative method. The stress, force in the string (under initial strain and fixed ends), and the first three natural frequencies of lateral vibration of the stretched string are determined. All dimensions are in inches.



**Figure 1. String Under Tension**

### Autodesk Nastran Analysis Model Filenames

- vm4\_8d.nas – Diagonal mass formulation
- vm4\_8c.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- 14 nodes, 13 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 100$  in

#### *Cross Sectional Properties*

Area:  $A = 3.06796 \text{ E-3 in}^2$

#### *Material Properties*

Young's Modulus:  $E = 30.0 \text{ E+3 ksi}$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 7.3 \text{ E-4 lb-sec}^2 / \text{in}^4$

### Boundary Conditions

The beam is restrained in all translations at end A, while end B has the Y and Z-translations restrained. All other nodes are restrained in the Z-translation, X and Y-rotations. The load is an initial strain  $e_0$  of 0.00543248 and a uniform mass along the string based on the mass density. This translates into a load  $P = \text{stress} \times \text{area} = E \times e_0 \times A = 500 \text{ lb}$ .

### Solution Type

Linear Prestress – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1, Table 2 and Table 3.

**Table 1. Force Results**

| Theory     | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Force (lb) | Force (lb)                                 | Error (%) | Force (lb)                                | Error (%) |
| 500        | 500                                        | 0.0       | 500                                       | 0.0       |

**Table 2. Stress Results**

| Theory       | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|--------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Stress (psi) | Stress (psi)                               | Error (%) | Stress (psi)                              | Error (%) |
| 162,974      | 162,973                                    | 0.0       | 162,973                                   | 0.0       |

**Table 3. Frequency Results**

|             | Theory                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 74.708                 | 74.645                                     | 0.1       | 74.708                                    | 0.0       |
| 2           | 149.42                 | 148.88                                     | 0.4       | 149.42                                    | 0.0       |
| 3           | 224.12                 | 222.18                                     | 0.9       | 224.13                                    | 0.0       |

### References

1. Thompson, W. T., *Vibration Theory and Applications*, 2<sup>nd</sup> Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.

## 4.9 Vibration of a Wedge

### Problem Description

Figure 1 shows the wedge plate model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The fundamental frequency of lateral vibration of the wedge plate is determined. All dimensions are in inches.

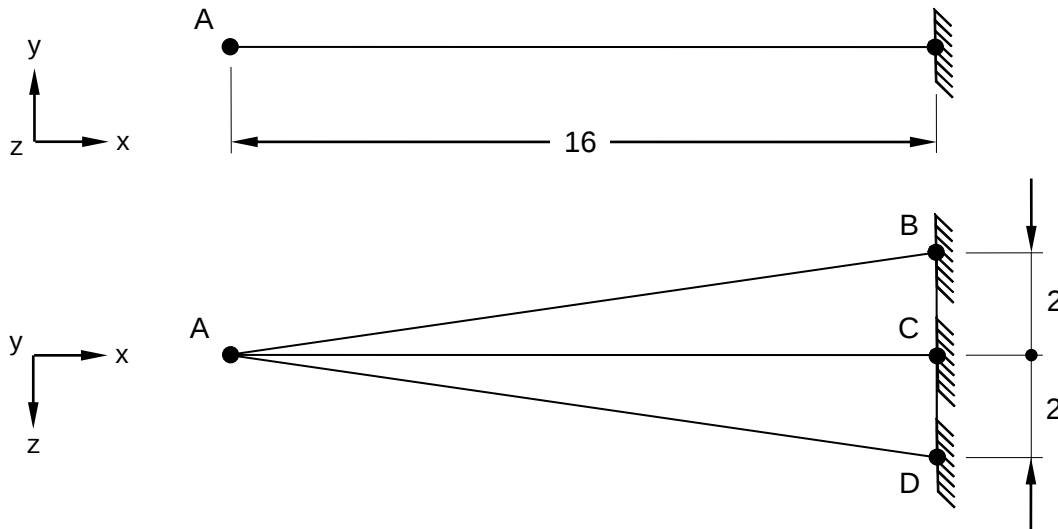


Figure 1. Wedge Plate

### Autodesk Nastran Analysis Model Filenames

- vm4\_9d.nas – Diagonal mass formulation
- vm4\_9c.nas – Coupled mass formulation

### Model Data

#### Finite Element Modeling

- 31 nodes, 38 5-DOF/node triangle elements

#### Units

inch/pound/second

#### Model Geometry

Length:  $L = 16$  in

Width:  $w = 2$  in

Thickness:  $t = 1$  in

### Material Properties

Young's Modulus:  $E = 30.0 \text{ E}+3 \text{ ksi}$

Shear Modulus of Elasticity:  $G = 1.5 \text{ E}+3 \text{ ksi}$

Mass Density:  $\rho = 7.28 \text{ E}-4 \text{ lb-sec}^2 / \text{in}^4$

### Boundary Conditions

At fixed end (edge BCD) the nodes are restrained in all translations and rotations. The load is a uniform mass over surface area based on mass density and material thickness.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|                | Theory                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1              | 259.16                       | 257.34                                        | 0.7       | 259.60                                       | 0.2       |

### References

1. Timoshenko, S. and Young, D. H., *Vibration Problems in Engineering*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955.

## 5. Normal Modes/Eigenvalue Verification Using Standard NAFEMS Benchmarks

The purpose of these normal mode dynamic test cases is to verify the functionality of Autodesk Nastran using standard benchmarks published by NAFEMS (National Agency for Finite Element Methods and Standards, National Engineering Laboratory, Glasgow, U.K.).

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

## 5.1 Bar Element Test Cases

The following normal mode/eigenvalue verification problems using standard NAFEMS Benchmarks are performed using bar elements.

### 5.1.1 Pin-ended Cross – In-plane Vibration

#### Problem Description

Figure 1 shows the pin-ended cross. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the pin-ended cross are determined. All dimensions are in meters.

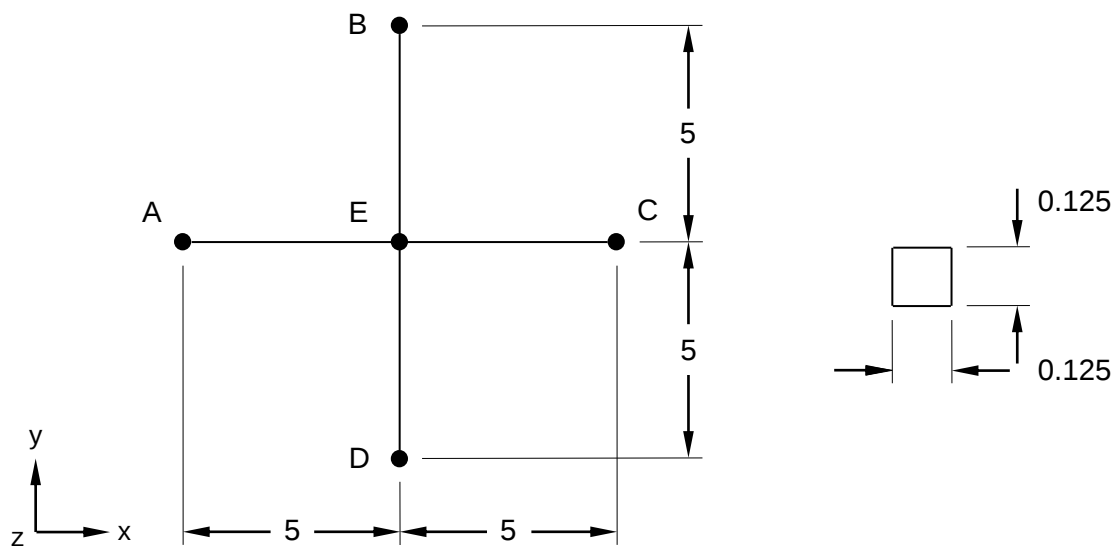


Figure 1. Pin-ended Cross

#### Autodesk Nastran Analysis Model Filenames

- vm5\_1\_1c.nas – 5-DOF/node, coupled mass formulation
- vm5\_1\_1d.nas – 5-DOF/node, diagonal mass formulation

#### Model Data

##### *Finite Element Modeling*

- Attributes: Coupling between flexural and extensional behavior. Repeated and closed eigenvalues.
- 17 nodes, 16 bar elements; four elements per arm (AE, BE, DE, CE)

##### *Units*

meter/Newton/second

**Model Geometry**

Length:  $L = 10 \text{ m}$

**Cross Sectional Properties**

Square Cross Section = (0.125 m x 0.125 m)

Area:  $A = 0.015625 \text{ m}^2$

Shear Ratio:

$Y = 0$

$Z = 0$

**Material Properties**

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Shear Modulus of Elasticity:  $G = 8.01 \text{ E}+10 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.29$

**Boundary Conditions**

Points A, B, C, D (nodes 2, 3, 4, 5) are constrained in all directions except for the Z-rotation. Point E (node 1, at coordinates 0,0,0) is constrained in the Z-translation and X-rotation. All other nodes (6–17) are constrained in the Z-translation and X and Y-rotations.

**Solution Type**

Normal Modes/Eigenvalue – Subspace iterative method

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

|                | NAFEMS                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1              | 11.34                        | 11.33                                         | 0.1       | 11.34                                        | 0.0       |
| 2, 3           | 17.69                        | 17.66                                         | 0.2       | 17.69                                        | 0.0       |
| 4              | 17.72                        | 17.69                                         | 0.2       | 17.72                                        | 0.0       |
| 5              | 45.48                        | 45.02                                         | 1.0       | 45.48                                        | 0.0       |
| 6, 7           | 57.36                        | 56.07                                         | 2.2       | 57.37                                        | 0.0       |
| 8              | 57.68                        | 56.35                                         | 2.3       | 57.69                                        | 0.0       |

**References**

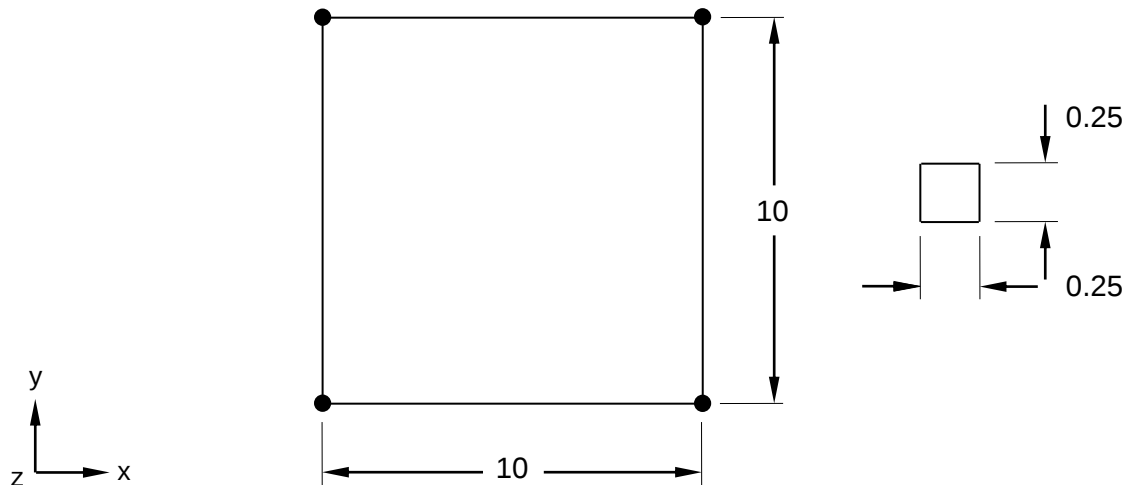
1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 1.



## 5.1.2 Free Square Frame – In-plane Vibration

### Problem Description

Figure 1 shows the free square frame. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square are determined. All dimensions are in meters.



**Figure 1. Free Square Frame**

### Autodesk Nastran Analysis Model Filenames

- vm5\_1\_2c.nas – 5-DOF/node, coupled mass formulation
- vm5\_1\_2d.nas – 5-DOF/node, diagonal mass formulation

### Model Data

#### *Finite Element Modeling*

- Attributes: Coupling between flexural and extensional behavior. Rigid body modes (3 modes). Repeated and close eigenvalues.
- 16 nodes, 16 bar elements; four elements per arm.

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10$  m

### Cross Sectional Properties

Square Cross Section = (0.25 m x 0.25 m)

Shear Ratio:

$Y = 1.0$

$Z = 1.0$

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

### Boundary Conditions

Constraint Set 1: All the nodes are constrained in the Z-translation and X and Y-rotations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|                | NAFEMS                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 4              | 3.262                        | 3.259                                         | 0.1       | 3.259                                        | 0.1       |
| 5              | 5.665                        | 5.660                                         | 0.1       | 5.662                                        | 0.1       |
| 6, 7           | 11.14                        | 10.89                                         | 2.3       | 11.13                                        | 0.1       |
| 8              | 12.83                        | 12.74                                         | 0.8       | 12.79                                        | 0.3       |
| 9              | 24.66                        | 23.53                                         | 4.6       | 24.61                                        | 0.2       |
| 10, 11         | 28.81                        | 28.13                                         | 2.3       | 28.70                                        | 0.4       |

### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 3.

### 5.1.3 Cantilever with Off-center Point Masses

#### Problem Description

Figure 1 shows the cantilever with off-center point masses. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the cantilever are determined. All dimensions are in meters.

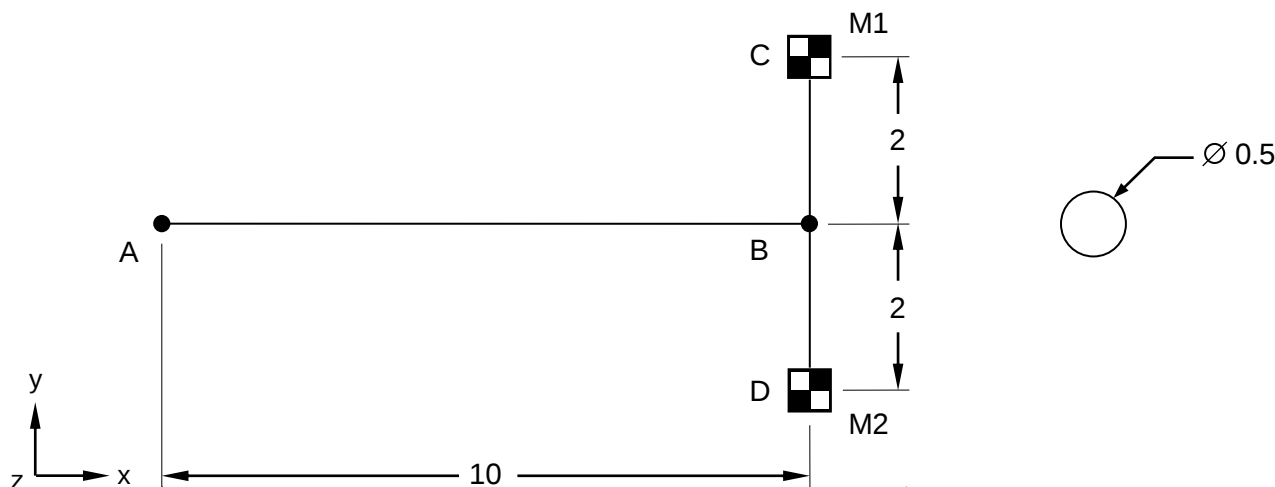


Figure 1. Cantilever with Off-center Point Mass

#### Autodesk Nastran Analysis Model Filenames

- vm5\_1\_3c.nas – 5-DOF/node, coupled mass formulation
- vm5\_1\_3d.nas – 5-DOF/node, diagonal mass formulation

#### Model Data

##### Finite Element Modeling

- Attributes: Coupling between torsional and flexural behavior. Internal axis non-coincident with flexibility axis. Discrete mass, rigid links. Close eigenvalues.
- 8 nodes, 9 elements; five bar elements along cantilever (from point A to point B, as shown in Figure 1), two mass elements (at point C,  $M1 = 10,000$  kg, and point D,  $M2 = 1,000$  kg), and two rigid elements (between points C and B, and points B and D, respectively).

##### Units

meter/Newton/second

##### Model Geometry

Length:  $L_1 = 10$  m and  $L_2 = 4$  m

### Cross Sectional Properties

Circular Cross Section = 0.5 m

Shear Ratio:

$Y = 1.128$

$Z = 1.128$

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Point A (node 1) is fully constrained in all translations and rotations (see Figure 1).

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|                | NAFEMS                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1              | 1.72                         | 1.71                                          | 0.6       | 1.72                                         | 0.0       |
| 2              | 1.73                         | 1.72                                          | 0.6       | 1.73                                         | 0.0       |
| 3              | 7.41                         | 7.56                                          | 2.0       | 7.55                                         | 1.9       |
| 4              | 9.97                         | 9.95                                          | 0.2       | 9.95                                         | 0.2       |
| 5              | 18.16                        | 17.68                                         | 2.6       | 18.07                                        | 0.5       |
| 6              | 26.97                        | 26.78                                         | 0.7       | 26.72                                        | 0.9       |

### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 4.

### 5.1.4 Deep Simply-Supported Beam

#### Problem Description

Figure 1 shows the deep simply supported beam. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the simply supported beam are determined. All dimensions are in meters.

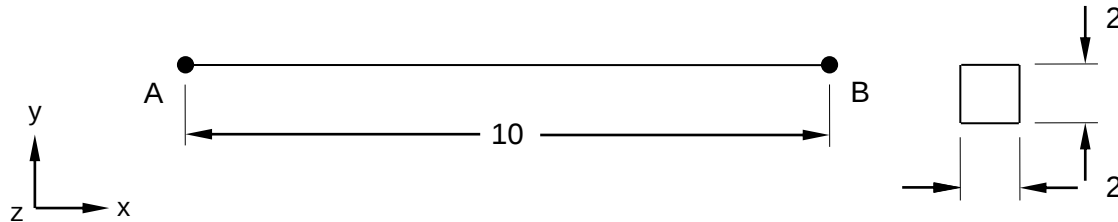


Figure 1. Deep Simply – Supported Beam

#### Model Input and FEMAP Neutral Base Filenames

- vm5\_1\_4c.nas – 5-DOF/node, coupled mass formulation
- vm5\_1\_4d.nas – 5-DOF/node, diagonal mass formulation

#### Model Data

##### Finite Element Modeling

- Attributes: Shear deformation and rotary inertial (Timoshenko beam). Possibility of missing extensional modes when using iteration solution methods. Repeated eigenvalues.
- 6 nodes, 5 bar elements (from point A to point B, as shown in Figure 1).

##### Units

meter/Newton/second

##### Model Geometry

Length:  $L = 10$  m

##### Cross Sectional Properties

Square Cross Section = (2 m x 2 m)

Shear Ratio:

$Y = 1.176923$

$Z = 1.176923$

##### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Point A (node 1) is constrained in the X, Y, and Z-translations and X-rotation. Point B (node 6) is constrained in the Y and Z-translations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|                | NAFEMS                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1, 2           | 42.57                        | 42.58                                         | 0.0       | 42.71                                        | 0.3       |
| 3              | 77.84                        | 77.19                                         | 0.8       | 77.83                                        | 0.0       |
| 4              | 125.51                       | 124.45                                        | 0.8       | 125.51                                       | 0.0       |
| 5, 6           | 145.46                       | 145.43                                        | 0.0       | 150.75                                       | 3.7       |
| 7              | 241.24                       | 224.02                                        | 7.1       | 241.20                                       | 0.0       |
| 8, 9           | 267.01                       | 263.73                                        | 1.2       | 301.07                                       | 12.8      |

### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 5.

### 5.1.5 Pin-ended Double Cross – In-plane Vibration

#### Problem Description

Figure 1 shows the pin-ended double cross. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The first 16 natural frequencies of vibration of the pin-ended double cross are determined. All dimensions are in meters.

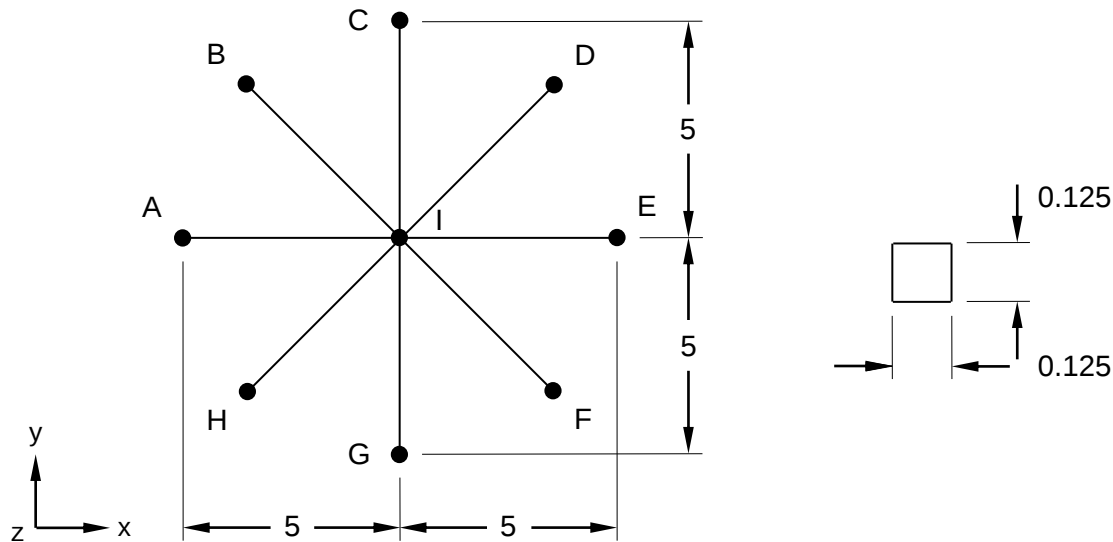


Figure 1. Pin-ended Double Cross

#### Autodesk Nastran Analysis Model Filenames

- vm5\_1\_5c.nas – 5-DOF/node, coupled mass formulation
- vm5\_1\_5d.nas – 5-DOF/node, diagonal mass formulation

#### Model Data

##### *Finite Element Modeling*

- Attributes: Coupling between flexural and extensional behavior. Repeated and closed eigenvalues.
- 33 nodes, 32 bar elements; four elements per arm.

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L = 10$  m

### Cross Sectional Properties

Square Cross Section = (0.125 m x 0.125 m)

Area:  $A = 0.015625 \text{ m}^2$

Moment of Inertia:  $I = 2.034\text{E-}5 \text{ m}^4$

### Material Properties

Young's Modulus:  $E = 20.0 \text{ E+}10 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.29$

### Boundary Conditions

Points A, B, C, D, E, F, G, H are constrained in all directions except for the Z-rotation. All other nodes are constrained in the Z-translation, X and Y-rotations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|                | NAFEMS                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode Number    | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1              | 11.336                       | 11.334                                        | 0.0       | 11.338                                       | 0.0       |
| 2, 3           | 17.709                       | 17.664                                        | 0.3       | 17.689                                       | 0.1       |
| 4,5,6,7,8      | 17.709                       | 17.693                                        | 0.1       | 17.717                                       | 0.0       |
| 9              | 45.345                       | 45.021                                        | 0.7       | 45.483                                       | 0.3       |
| 10,11          | 57.390                       | 56.065                                        | 2.3       | 57.371                                       | 0.0       |
| 12,13,14,15,16 | 57.390                       | 56.351                                        | 1.8       | 57.690                                       | 0.5       |

### References

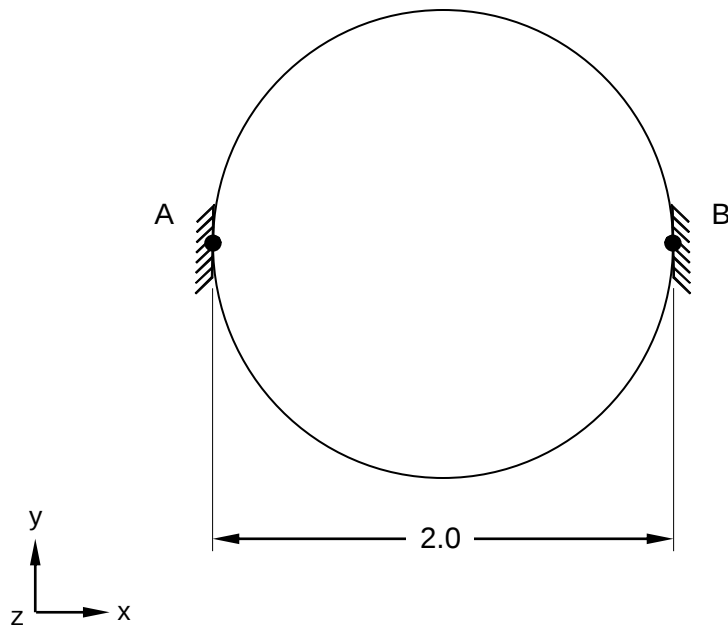
1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 1.



### 5.1.6 Circular Ring – In-plane and Out-of-plane Vibration

#### Problem Description

Figure 1 shows the circular ring. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The first 16 natural frequencies of vibration of the circular ring are determined. All dimensions are in meters.



**Figure 1. Circular Ring**

#### Autodesk Nastran Analysis Model Filenames

- vm5\_1\_6c.nas – 5-DOF/node, coupled mass formulation
- vm5\_1\_6d.nas – 5-DOF/node, diagonal mass formulation

#### Model Data

##### *Finite Element Modeling*

- Attributes: Rigid Body Modes. Repeated and closed eigenvalues.
- 20 nodes, 20 bar elements

##### *Units*

meter/Newton/second

##### *Model Geometry*

Radius:  $R = 1$  m

### Cross Sectional Properties

Area:  $A = 0.00785398 \text{ m}^2$  ;  $A_y = A_z = 0.00696148 \text{ m}^2$

Moment of Inertia:  $I = 4.90874\text{E-}6 \text{ m}^4$

Torsional Constant:  $J = 9.81748\text{E-}6 \text{ m}^4$

### Material Properties

Young's Modulus:  $E = 20.0 \text{ E+}10 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Points A and B are constrained in all translations and rotations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

|                      | NAFEMS                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode Number          | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 7,8 (out of plane)   | 51.849                       | 51.617                                        | 0.4       | 52.210                                       | 0.7       |
| 9,10 (in plane)      | 53.382                       | 54.054                                        | 1.3       | 53.776                                       | 0.7       |
| 11,12 (out of plane) | 148.77                       | 146.94                                        | 1.2       | 148.92                                       | 0.1       |
| 13,14 (in plane)     | 150.99                       | 152.21                                        | 0.8       | 151.26                                       | 0.2       |
| 15 (out of plane)    | 286.98                       | 280.42                                        | 2.3       | 285.33                                       | 0.6       |
| 16 (out of plane)    | 289.51                       | 280.42                                        | 2.3       | 285.33                                       | 0.6       |

### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 6.

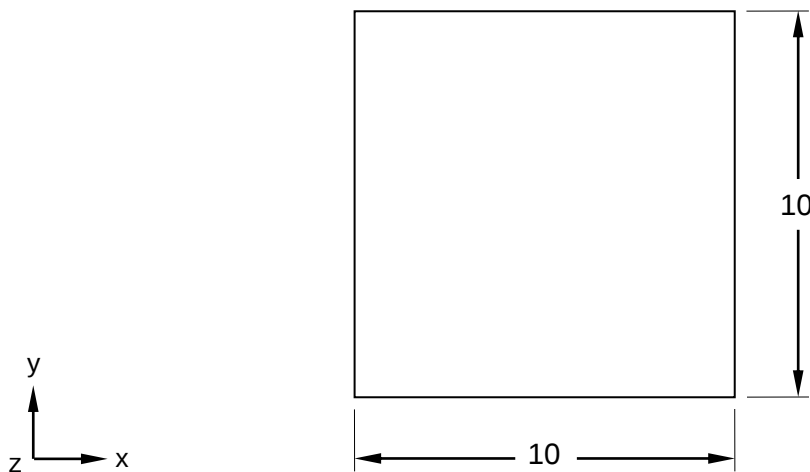
## 5.2 Plate Element Test Cases

The following normal mode/eigenvalue verification problems using standard NAFEMS Benchmarks are performed using plate elements.

### 5.2.1 Thin Square Cantilevered Plate – Symmetric Modes

#### Problem Description

Figure 1 shows the thin square cantilevered plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.



**Figure 1. Thin Square Cantilevered Plate**

#### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_1c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5\_2\_1d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5\_2\_1cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5\_2\_1dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

#### Model Data

##### *Finite Element Modeling*

- Attributes: Symmetric modes, symmetric boundary conditions along the cutting plane.
- Test 1 and 2 (vm5\_2\_1c4 and vm5\_2\_1d4): 45 nodes, 32 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_1cR and vm5\_2\_1dR): 45 nodes, 32 6-DOF/node quadrilateral plate elements.
- Only the bottom half of the plate is meshed (10 m x 5 m).
- There are 4 elements vertically, on the 5 m portion of the plate and 8 elements horizontally, on the 10 m portion of the plate.

#### *Units*

meter/Newton/second

**Model Geometry**

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m

**Material Properties**

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

Constraints (all tests): The model is constrained using symmetry boundary conditions. The nodes on the left hand side edge are fully constrained in all translations and rotations. All other nodes are constrained in the X and Y-translations and the Z-rotation.

**Solution Type**

Normal Modes/Eigenvalue – Subspace iterative method

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | Ref. Value             | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 0.421                  | 0.416                                      | 1.2       | 0.419                                     | 0.5       |
|             | CQUADR  | 0.421                  | 0.416                                      | 1.2       | 0.419                                     | 0.5       |
| 2           | CQUAD4  | 2.582                  | 2.496                                      | 3.3       | 2.611                                     | 1.1       |
|             | CQUADR  | 2.582                  | 2.496                                      | 3.3       | 2.611                                     | 1.1       |
| 3           | CQUAD4  | 3.306                  | 3.152                                      | 4.6       | 3.354                                     | 1.4       |
|             | CQUADR  | 3.306                  | 3.152                                      | 4.6       | 3.354                                     | 1.4       |
| 4           | CQUAD4  | 6.555                  | 6.267                                      | 4.4       | 6.894                                     | 5.2       |
|             | CQUADR  | 6.555                  | 6.267                                      | 4.4       | 6.894                                     | 5.2       |
| 5           | CQUAD4  | 7.381                  | 7.156                                      | 3.0       | 7.882                                     | 6.3       |
|             | CQUADR  | 7.381                  | 7.156                                      | 3.0       | 7.882                                     | 6.3       |
| 6           | CQUAD4  | 11.402                 | 11.253                                     | 1.3       | 13.164                                    | 15.4      |
|             | CQUADR  | 11.402                 | 11.253                                     | 1.3       | 13.164                                    | 15.4      |

**Note:** Reference value (Ref. Value) refers to the accepted solution to this problem.

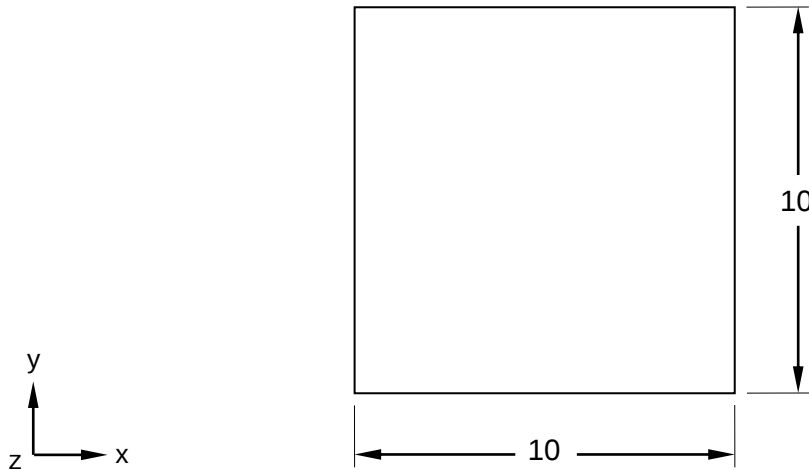
## References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 11a.

## 5.2.2 Thin Square Cantilevered Plate – Anti-symmetric Modes

### Problem Description

Figure 1 shows the thin square cantilevered plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.



**Figure 1. Thin Square Cantilevered Plate**

### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_2c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5\_2\_2d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5\_2\_2cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5\_2\_2dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

### Model Data

#### *Finite Element Modeling*

- Attributes: Anti-symmetric modes.
- Test 1 and 2 (vm5\_2\_2c4 and vm5\_2\_2d4): 45 nodes, 32 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_2cR and vm5\_2\_2dR): 45 nodes, 32 6-DOF/node quadrilateral plate elements.
- Only the bottom half of the plate is meshed (10 m x 5 m).
- There are 4 elements vertically, on the 5 m portion of the plate, and 8 elements horizontally, on the 10 m portion of the plate.

### *Units*

meter/Newton/second

### Model Geometry

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Constraints (all tests): Nodes 1-5 are fully constrained in all translations and rotations. Nodes 6, 11, 16, 21, 26, 31, 36, and 41 are constrained in the X, Y, and Z-translations and Z-rotation. All other nodes are constrained in the X and Y-translations and Z-rotation.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 1.019                  | 1.012                                      | 0.7       | 1.032                                     | 1.3       |
|             | CQUADR  | 1.019                  | 1.012                                      | 0.7       | 1.032                                     | 1.3       |
| 2           | CQUAD4  | 3.839                  | 3.641                                      | 5.1       | 3.843                                     | 0.1       |
|             | CQUADR  | 3.839                  | 3.641                                      | 5.1       | 3.843                                     | 0.1       |
| 3           | CQUAD4  | 8.313                  | 7.199                                      | 13.4      | 8.248                                     | 0.8       |
|             | CQUADR  | 8.313                  | 7.199                                      | 13.4      | 8.248                                     | 0.8       |
| 4           | CQUAD4  | 9.424                  | 8.303                                      | 11.9      | 9.321                                     | 1.1       |
|             | CQUADR  | 9.424                  | 8.303                                      | 11.9      | 9.321                                     | 1.1       |
| 5           | CQUAD4  | 11.728                 | 10.411                                     | 11.2      | 12.275                                    | 4.7       |
|             | CQUADR  | 11.728                 | 10.411                                     | 11.2      | 12.275                                    | 4.7       |
| 6           | CQUAD4  | 17.818                 | 15.097                                     | 15.2      | 18.366                                    | 3.1       |
|             | CQUADR  | 17.818                 | 15.097                                     | 15.2      | 18.366                                    | 3.1       |

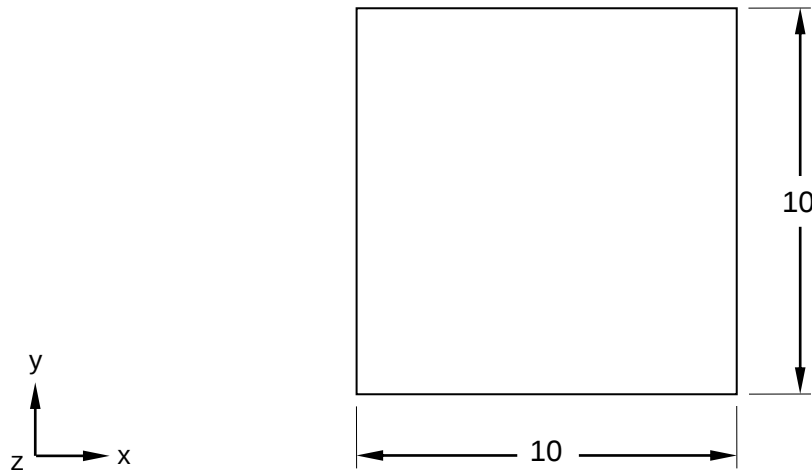
### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 11b.

### 5.2.3 Free Thin Square Plate

#### Problem Description

Figure 1 shows the thin square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.



**Figure 1. Free Thin Square Plate**

#### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_3c4.nas – 5-DOF/node, CQUAD4, coupled mass
- vm5\_2\_3d4.nas – 5-DOF/node, CQUAD4, diagonal mass
- vm5\_2\_3cR.nas – 6-DOF/node, CQUADR, coupled mass
- vm5\_2\_3dR.nas – 6-DOF/node, CQUADR, diagonal mass

#### Model Data

##### *Finite Element Modeling*

- Attributes: Rigid body modes (three modes). Repeated eigenvalues. Use of kinematic DOF for the rigid body mode calculation with the subspace iterative eigensolver.
- Test 1 and 2 (vm5\_2\_3c4 and vm5\_2\_3d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_3cR and vm5\_2\_3dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The whole plate is modeled with 8 elements on each side.

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m



### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Constraint set 1: All nodes are constrained in the X and Y-translations and Z-rotation.

Constraint set 2: Nodes 1 and 3 (lower left and upper right) are fully constrained in all translations and rotations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | Ref. Value             | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 4           | CQUAD4  | 1.622                  | 1.587                                      | 2.1       | 1.633                                     | 0.8       |
|             | CQUADR  | 1.622                  | 1.587                                      | 2.1       | 1.633                                     | 0.8       |
| 5           | CQUAD4  | 2.360                  | 2.242                                      | 5.0       | 2.390                                     | 1.3       |
|             | CQUADR  | 2.360                  | 2.242                                      | 5.0       | 2.390                                     | 1.3       |
| 6           | CQUAD4  | 2.922                  | 2.804                                      | 4.0       | 2.978                                     | 1.9       |
|             | CQUADR  | 2.922                  | 2.804                                      | 4.0       | 2.978                                     | 1.9       |
| 7, 8        | CQUAD4  | 4.233                  | 3.980                                      | 6.0       | 4.293                                     | 1.4       |
|             | CQUADR  | 4.233                  | 3.980                                      | 6.0       | 4.293                                     | 1.4       |
| 9           | CQUAD4  | 7.416                  | 6.844                                      | 7.7       | 7.816                                     | 5.4       |
|             | CQUADR  | 7.416                  | 6.844                                      | 7.7       | 7.816                                     | 5.4       |
| 10          | CQUAD4  | N/A                    | 6.844                                      | N/A       | 7.816                                     | N/A       |
|             | CQUADR  | N/A                    | 6.844                                      | N/A       | 7.816                                     | N/A       |

**Note:** Reference value (Ref. Value) refers to the accepted solution to this problem.

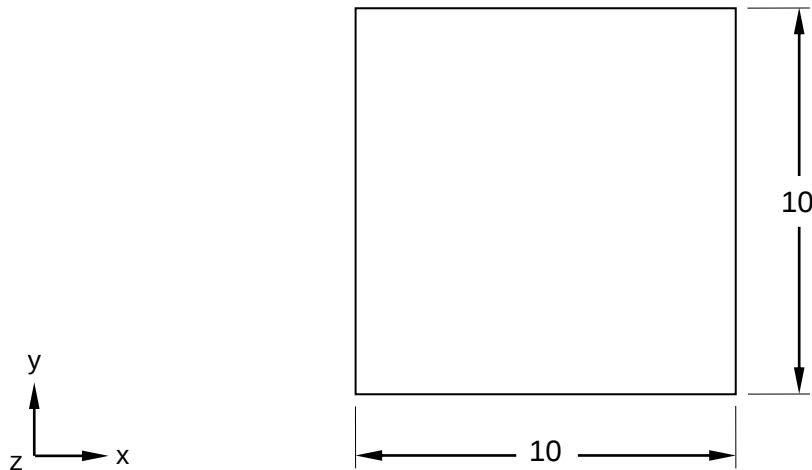
### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 12.

## 5.2.4 Simply – Supported Thin Square Plate

### Problem Description

Figure 1 shows the simply – supported thin square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.



**Figure 1. Simply – Supported Thin Square Plate**

### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_4c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5\_2\_4d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5\_2\_4cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5\_2\_4dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

### Model Data

#### *Finite Element Modeling*

- Attributes: Well established, repeated eigenvalues.
- Test 1 and 2 (vm5\_2\_4c4 and vm5\_2\_4d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_4cR and vm5\_2\_4dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The whole plate is modeled with 8 elements on each edge.

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

All nodes are constrained in the X and Y-translations, and Z-rotation. All nodes along edges  $X = 0$  and  $X = 10 \text{ m}$  are constrained in the Z-translation and X-rotation, while the nodes along edges  $Y = 0$  and  $Y = 10 \text{ m}$  are constrained in the Z-translation and Y-rotation. A constraint set is created to fully constrain the four corner nodes (9, 13, 41, 68) in all translations and rotations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | Ref. Value             | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 2.377                  | 2.377                                      | 0.0       | 2.377                                     | 0.0       |
|             | CQUADR  | 2.377                  | 2.377                                      | 0.0       | 2.377                                     | 0.0       |
| 2, 3        | CQUAD4  | 5.942                  | 6.001                                      | 1.0       | 6.001                                     | 1.0       |
|             | CQUADR  | 5.942                  | 6.001                                      | 1.0       | 6.001                                     | 1.0       |
| 4           | CQUAD4  | 9.507                  | 9.526                                      | 0.2       | 9.527                                     | 0.2       |
|             | CQUADR  | 9.507                  | 9.526                                      | 0.2       | 9.527                                     | 0.2       |
| 5, 6        | CQUAD4  | 11.884                 | 12.325                                     | 3.7       | 12.327                                    | 3.7       |
|             | CQUADR  | 11.884                 | 12.325                                     | 3.7       | 12.327                                    | 3.7       |
| 7, 8        | CQUAD4  | 15.449                 | 15.756                                     | 2.0       | 15.758                                    | 2.0       |
|             | CQUADR  | 15.449                 | 15.756                                     | 2.0       | 15.758                                    | 2.0       |

**Note:** Reference value (Ref. Value) refers to the accepted solution to this problem.

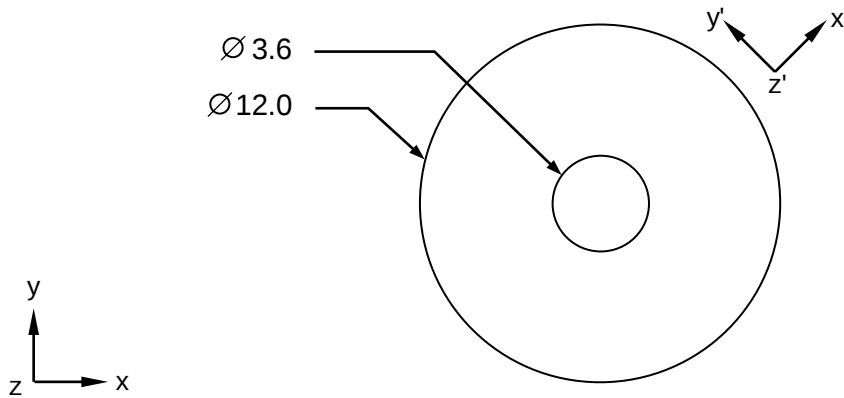
### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 13.

## 5.2.5 Simply – Supported Thin Annular Plate

### Problem Description

Figure 1 shows the simply – supported thin annular plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the annular plate are determined. All dimensions are in meters.



**Figure 1. Simply – Supported Thin Annular Plate**

### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_5c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5\_2\_5d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5\_2\_5cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5\_2\_5dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

### Model Data

#### *Finite Element Modeling*

- Attributes: Curved boundary (Skewed coordinate system). Repeated eigenvalues.
- Test 1 and 2 (vm5\_2\_5c4 and vm5\_2\_5d4): 192 nodes, 160 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_5cR and vm5\_2\_5dR): 192 nodes, 160 6-DOF/node quadrilateral plate elements.
- The model has a mapped mesh with 32 elements on the inner and outer circles of the plate, respectively.

#### *Units*

meter/Newton/second

#### *Model Geometry*

Outer Radius:  $R_o = 6.0$  m

Inner Radius:  $R_i = 1.8$  m

Thickness:  $t = 0.05$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Constraint set 1 (All Tests): All nodes are constrained in the R and T-translations, and Z-rotation. All nodes around the model's circumference are additionally constrained in the Z-translation and R-rotation.

Constraint set 2 (All Tests): Nodes 258 and 290 on the outer circumference are constrained in the R and T-translations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | Ref. Value             | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 1.870                  | 1.863                                      | 0.4       | 1.882                                     | 0.6       |
|             | CQUADR  | 1.870                  | 1.863                                      | 0.4       | 1.882                                     | 0.6       |
| 2, 3        | CQUAD4  | 5.137                  | 5.164                                      | 0.5       | 5.238                                     | 2.0       |
|             | CQUADR  | 5.137                  | 5.164                                      | 0.5       | 5.238                                     | 2.0       |
| 4, 5        | CQUAD4  | 9.673                  | 9.796                                      | 1.3       | 10.093                                    | 4.3       |
|             | CQUADR  | 9.673                  | 9.796                                      | 1.3       | 10.093                                    | 4.3       |
| 6           | CQUAD4  | 14.850                 | 14.239                                     | 4.1       | 15.469                                    | 4.2       |
|             | CQUADR  | 14.850                 | 14.239                                     | 4.1       | 15.469                                    | 4.2       |
| 7, 8        | CQUAD4  | 15.573                 | 15.743                                     | 1.1       | 16.606                                    | 6.6       |
|             | CQUADR  | 15.573                 | 15.743                                     | 1.1       | 16.606                                    | 6.6       |
| 9           | CQUAD4  | 18.382                 | 17.765                                     | 3.4       | 19.275                                    | 4.9       |
|             | CQUADR  | 18.382                 | 17.765                                     | 3.4       | 19.275                                    | 4.9       |

**Note:** Reference value (Ref. Value) refers to the accepted solution to this problem.

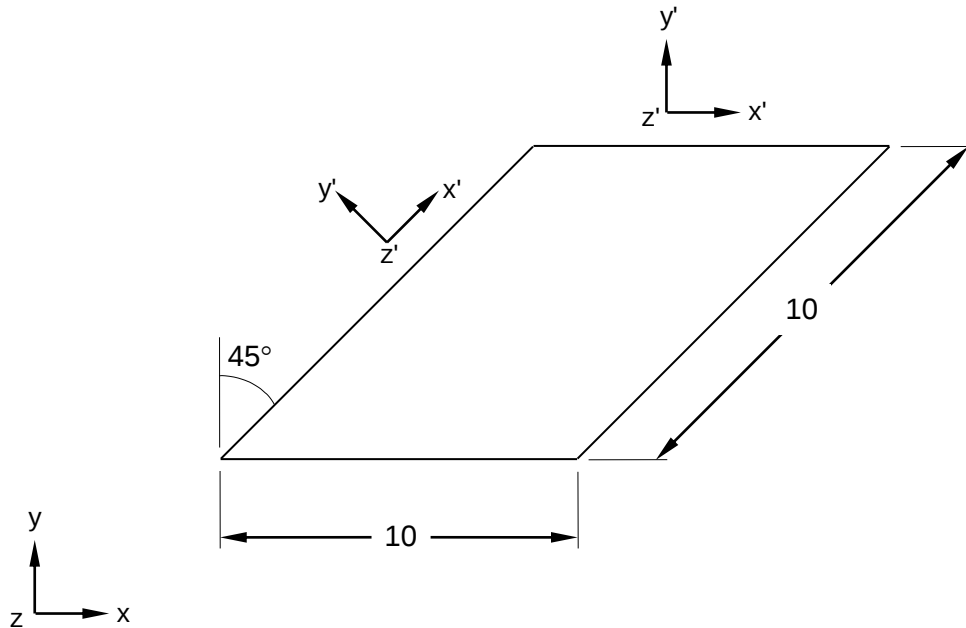
### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 14.

## 5.2.6 Clamped Thin Rhombic Plate

### Problem Description

Figure 1 shows the thin rhombic plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the rhombic plate are determined. All dimensions are in meters.



**Figure 1. Clamped Thin Rhombic Plate**

### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_6c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5\_2\_6d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5\_2\_6cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5\_2\_6dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

### Model Data

#### *Finite Element Modeling*

- Attributes: Distorted elements.
- Test 1 and 2 (vm5\_2\_6c4 and vm5\_2\_6d4): 169 nodes, 144 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_6cR and vm5\_2\_6dR): 169 nodes, 144 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 12 elements on each edge.

#### *Units*

meter/Newton/second

### Model Geometry

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The nodes along all four edges are fully constrained in all translations and rotations. All other nodes are constrained in the X and Y-translations, and Z-rotation.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 8.142                  | 7.988                                      | 1.9       | 8.130                                     | 0.1       |
|             | CQUADR  | 8.142                  | 7.988                                      | 1.9       | 8.130                                     | 0.1       |
| 2           | CQUAD4  | 13.891                 | 13.088                                     | 5.8       | 13.666                                    | 1.6       |
|             | CQUADR  | 13.891                 | 13.088                                     | 5.8       | 13.666                                    | 1.6       |
| 3           | CQUAD4  | 20.036                 | 18.443                                     | 8.0       | 19.776                                    | 1.3       |
|             | CQUADR  | 20.036                 | 18.443                                     | 8.0       | 19.776                                    | 1.3       |
| 4           | CQUAD4  | 20.165                 | 19.340                                     | 4.1       | 20.071                                    | 0.5       |
|             | CQUADR  | 20.165                 | 19.340                                     | 4.1       | 20.071                                    | 0.5       |
| 5           | CQUAD4  | 27.704                 | 24.901                                     | 10.1      | 27.582                                    | 0.4       |
|             | CQUADR  | 27.704                 | 24.901                                     | 10.1      | 27.582                                    | 0.4       |
| 6           | CQUAD4  | 32.046                 | 28.777                                     | 10.2      | 31.039                                    | 3.1       |
|             | CQUADR  | 32.046                 | 28.777                                     | 10.2      | 31.039                                    | 3.1       |

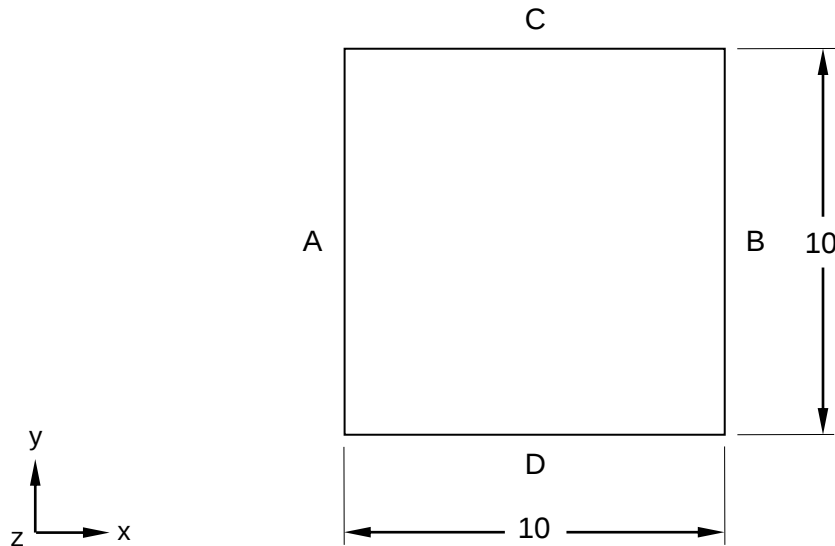
### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 15.

## 5.2.7 Simply – Supported Thick Square Plate, Test A

### Problem Description

Figure 1 shows the thick square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.



**Figure 1. Simply – Supported Thick Square Plate**

### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_7c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5\_2\_7d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5\_2\_7cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5\_2\_7dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

### Model Data

#### *Finite Element Modeling*

- Attributes: Well established. Repeated eigenvalues. Effect of secondary restraints.
- Test 1 and 2 (vm5\_2\_7c4 and vm5\_2\_7d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_7cR and vm5\_2\_7dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 8 elements on each edge.

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m



### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The corner nodes are fully constrained in all translations and rotations. The nodes along edges A and B (see Figure 1) are constrained in all translations and rotations, except the Y-rotation. The nodes along edges C and D are constrained in all translations and rotations, except the X-rotation. All other nodes are constrained in the X and Y-translations, and Z-rotation.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 46.66                  | 46.21                                      | 1.0       | 47.06                                     | 0.8       |
|             | CQUADR  | 46.66                  | 46.21                                      | 1.0       | 47.06                                     | 0.8       |
| 2, 3        | CQUAD4  | 115.84                 | 111.00                                     | 4.2       | 116.47                                    | 0.6       |
|             | CQUADR  | 115.84                 | 111.00                                     | 4.2       | 116.47                                    | 0.6       |
| 4           | CQUAD4  | 177.53                 | 169.48                                     | 4.5       | 183.63                                    | 3.4       |
|             | CQUADR  | 177.53                 | 169.48                                     | 4.5       | 183.63                                    | 3.4       |
| 5, 6        | CQUAD4  | 233.40                 | 208.10                                     | 10.8      | 230.76                                    | 1.1       |
|             | CQUADR  | 233.40                 | 208.10                                     | 10.8      | 230.76                                    | 1.1       |
| 7, 8        | CQUAD4  | 283.60                 | 256.70                                     | 9.5       | 294.72                                    | 3.9       |
|             | CQUADR  | 283.60                 | 256.70                                     | 9.5       | 294.72                                    | 3.9       |
| 9, 10       | CQUAD4  | 371.11                 | 322.76                                     | 13.0      | 389.14                                    | 4.8       |
|             | CQUADR  | 371.11                 | 322.76                                     | 13.0      | 389.14                                    | 4.8       |

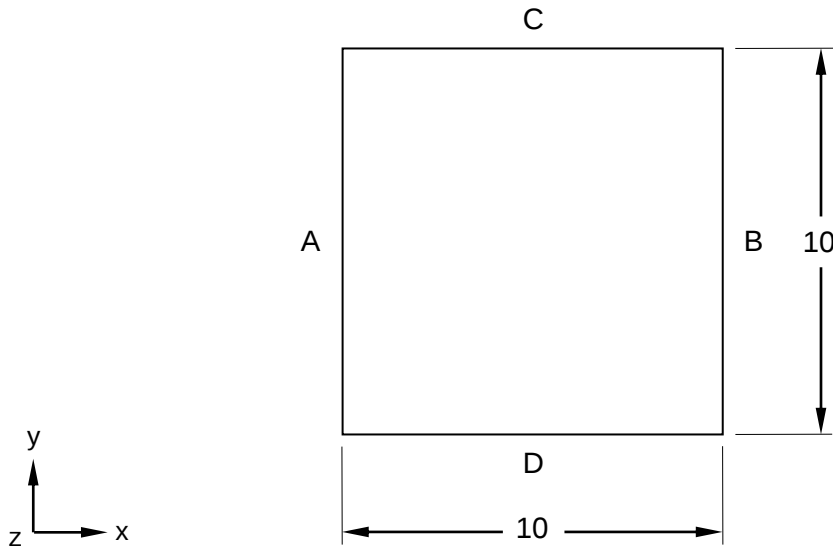
### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 21a.

## 5.2.8 Simply – Supported Thick Square Plate, Test B

### Problem Description

Figure 1 shows the thick square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.



**Figure 1. Simply – Supported Thick Square Plate**

### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_8c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5\_2\_8d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5\_2\_8cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5\_2\_8dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

### Model Data

#### *Finite Element Modeling*

- Attributes: Well established. Repeated eigenvalues. Effect of secondary restraints.
- Test 1 and 2 (vm5\_2\_8c4 and vm5\_2\_8d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_8cR and vm5\_2\_8dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 8 elements on each edge.

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The corner nodes are fully constrained in all translations and rotations. The nodes along edges A and B are constrained in all translations and rotations, except the Y-rotation (see Figure 1). The nodes along edges C and D are constrained in all translations and rotations, except the X-rotation. All other nodes are constrained in the X and Y-translations, and Z-rotation.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 44.75                  | 44.36                                      | 0.9       | 45.510                                    | 1.7       |
|             | CQUADR  | 44.75                  | 44.36                                      | 0.9       | 45.510                                    | 1.7       |
| 2, 3        | CQUAD4  | 112.94                 | 106.97                                     | 5.3       | 114.04                                    | 0.9       |
|             | CQUADR  | 112.94                 | 106.97                                     | 5.3       | 114.04                                    | 0.9       |
| 4           | CQUAD4  | 170.28                 | 160.34                                     | 5.9       | 177.56                                    | 4.3       |
|             | CQUADR  | 170.28                 | 160.34                                     | 5.9       | 177.56                                    | 4.3       |
| 5, 6        | CQUAD4  | 230.23                 | 200.62                                     | 12.9      | 227.94                                    | 0.8       |
|             | CQUADR  | 230.23                 | 200.62                                     | 12.9      | 227.94                                    | 0.8       |
| 7, 8        | CQUAD4  | 274.19                 | 242.87                                     | 11.4      | 286.52                                    | 4.5       |
|             | CQUADR  | 274.19                 | 242.87                                     | 11.4      | 286.52                                    | 4.5       |
| 9           | CQUAD4  | 355.98                 | 308.88                                     | 13.2      | 386.48                                    | 8.6       |
|             | CQUADR  | 355.98                 | 308.88                                     | 13.2      | 386.48                                    | 8.6       |
| 10          | CQUAD4  | 355.98                 | 311.69                                     | 12.4      | 386.48                                    | 8.6       |
|             | CQUADR  | 355.98                 | 311.69                                     | 12.4      | 386.48                                    | 8.6       |

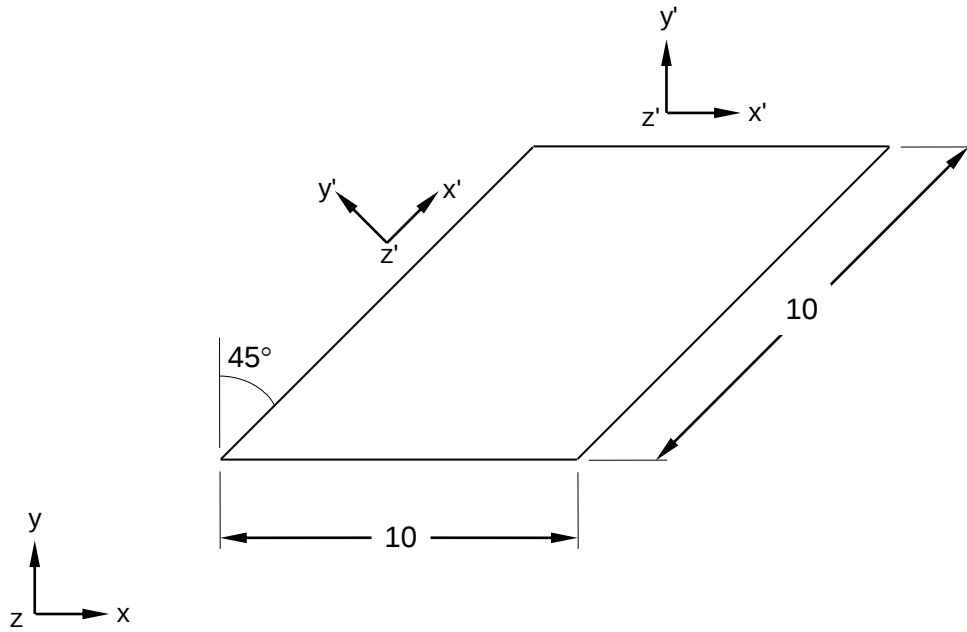
### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 21b.

## 5.2.9 Clamped Thick Rhombic Plate

### Problem Description

Figure 1 shows the thick rhombic plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the rhombic plate are determined. All dimensions are in meters.



**Figure 1. Clamped Thick Rhombic Plate**

### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_9c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5\_2\_9d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5\_2\_9cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5\_2\_9dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

### Model Data

#### *Finite Element Modeling*

- Attributes: Distorted elements.
- Test 1 and 2 (vm5\_2\_9c4 and vm5\_2\_9d4): 121 nodes, 100 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_9cR and vm5\_2\_9dR): 121 nodes, 100 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 10 elements on each edge.

### *Units*

meter/Newton/second

**Model Geometry**

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m

**Material Properties**

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

The nodes along all four edges are fully constrained in all translations and rotations. All other nodes are constrained in the X and Y-translations, and Z-rotation.

**Solution Type**

Normal Modes/Eigenvalue – Subspace iterative method

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 137.80                 | 133.46                                     | 3.1       | 136.71                                    | 0.8       |
|             | CQUADR  | 137.80                 | 133.46                                     | 3.1       | 136.71                                    | 0.8       |
| 2           | CQUAD4  | 218.48                 | 202.48                                     | 7.3       | 214.62                                    | 1.8       |
|             | CQUADR  | 218.48                 | 202.48                                     | 7.3       | 214.62                                    | 1.8       |
| 3           | CQUAD4  | 295.42                 | 265.73                                     | 10.1      | 291.70                                    | 1.3       |
|             | CQUADR  | 295.42                 | 265.73                                     | 10.1      | 291.70                                    | 1.3       |
| 4           | CQUAD4  | 296.83                 | 279.28                                     | 5.9       | 293.71                                    | 1.0       |
|             | CQUADR  | 296.83                 | 279.28                                     | 5.9       | 293.71                                    | 1.0       |
| 5           | CQUAD4  | 383.56                 | 331.75                                     | 13.5      | 380.35                                    | 0.8       |
|             | CQUADR  | 383.56                 | 331.75                                     | 13.5      | 380.35                                    | 0.8       |
| 6           | CQUAD4  | 426.59                 | 376.13                                     | 11.8      | 416.71                                    | 2.3       |
|             | CQUADR  | 426.59                 | 376.13                                     | 11.8      | 416.71                                    | 2.3       |

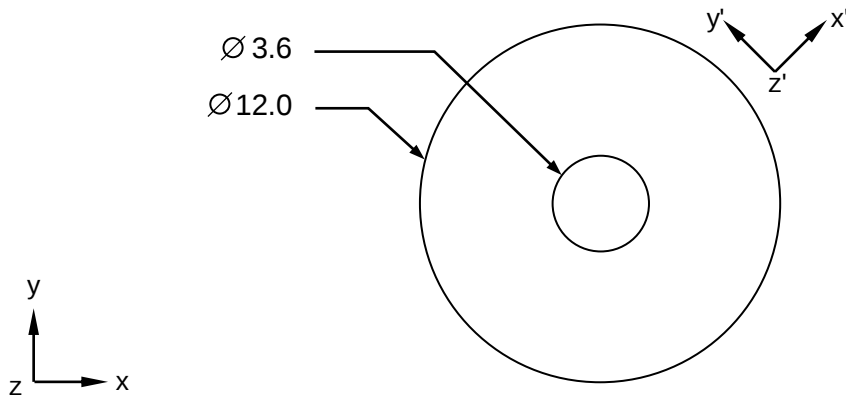
**References**

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 22.

### 5.2.10 Simply – Supported Thick Annular Plate

#### Problem Description

Figure 1 shows the thick annular plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the annular plate are determined. All dimensions are in meters.



**Figure 1. Simply – Supported Thick Annular Plate**

#### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_10c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm5\_2\_10d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm5\_2\_10cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation
- vm5\_2\_10dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation

#### Model Data

##### *Finite Element Modeling*

- Attributes: Curved boundary (skewed coordinate system). Repeated eigenvalues.
- Test 1 and 2 (vm5\_2\_10c4 and vm5\_2\_10d4): 192 nodes, 160 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_10cR and vm5\_2\_10dR): 192 nodes, 160 6-DOF/node quadrilateral plate elements.
- The model has a mapped mesh with 32 elements on the inner and outer circles of the plate, respectively.

##### *Units*

meter/Newton/second

##### *Model Geometry*

Outer Radius:  $R_o = 6.0$  m

Inner Radius:  $R_i = 1.8$  m

Thickness:  $t = 0.05$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The nodes around the circumference are constrained in the X, Y and Z-translations, and X and Z-rotations. All other nodes are constrained in the X and Y-translations, and Z-rotation.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 18.82                  | 18.50                                      | 1.7       | 18.68                                     | 0.7       |
|             | CQUADR  | 18.82                  | 18.50                                      | 1.7       | 18.68                                     | 0.7       |
| 2, 3        | CQUAD4  | 49.82                  | 49.44                                      | 0.8       | 50.13                                     | 0.6       |
|             | CQUADR  | 49.82                  | 49.44                                      | 0.8       | 50.13                                     | 0.6       |
| 4, 5        | CQUAD4  | 96.06                  | 93.46                                      | 2.7       | 96.26                                     | 0.2       |
|             | CQUADR  | 96.06                  | 93.46                                      | 2.7       | 96.26                                     | 0.2       |
| 6           | CQUAD4  | 148.34                 | 134.61                                     | 9.3       | 145.86                                    | 1.6       |
|             | CQUADR  | 148.34                 | 134.61                                     | 9.3       | 145.86                                    | 1.6       |
| 7, 8        | CQUAD4  | 153.68                 | 146.67                                     | 4.5       | 154.71                                    | 0.7       |
|             | CQUADR  | 153.68                 | 146.67                                     | 4.5       | 154.71                                    | 0.7       |
| 9           | CQUAD4  | 174.52                 | 161.03                                     | 7.7       | 174.52                                    | 0.0       |
|             | CQUADR  | 174.52                 | 161.03                                     | 7.7       | 174.52                                    | 0.0       |

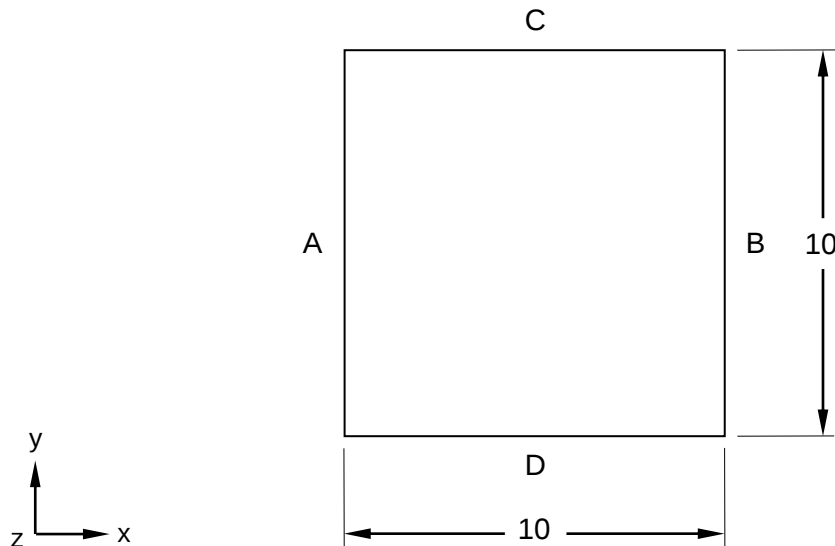
### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 23.

### 5.2.11 Cantilevered Square Membrane

#### Problem Description

Figure 1 shows the square membrane. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square membrane are determined. All dimensions are in meters.



**Figure 1. Cantilevered Square Membrane**

#### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_11c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm5\_2\_11d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm5\_2\_11cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation
- vm5\_2\_11dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation

#### Model Data

##### *Finite Element Modeling*

- Attributes: Well established.
- Test 1 and 2 (vm5\_2\_11c4 and vm5\_2\_11d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_11cR and vm5\_2\_11dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 8 elements on each edge.

##### *Units*

meter/Newton/second



### Model Geometry

Length:  $L = 10.0$  m

Thickness:  $t = 0.05$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The nodes along the Y-axis are fully constrained in all translations and rotations (edge A in Figure 1). All other nodes are constrained in the Z-translation, and all rotations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 52.90                  | 53.31                                      | 0.8       | 53.60                                     | 1.3       |
|             | CQUADR  | 52.90                  | 53.36                                      | 0.9       | 53.66                                     | 1.4       |
| 2           | CQUAD4  | 126.11                 | 125.60                                     | 0.4       | 126.07                                    | 0.0       |
|             | CQUADR  | 126.11                 | 125.62                                     | 0.4       | 126.09                                    | 0.0       |
| 3           | CQUAD4  | 143.20                 | 140.68                                     | 1.8       | 143.98                                    | 0.5       |
|             | CQUADR  | 143.20                 | 140.81                                     | 1.7       | 144.11                                    | 0.6       |
| 4           | CQUAD4  | 228.85                 | 216.41                                     | 5.4       | 228.76                                    | 0.0       |
|             | CQUADR  | 228.85                 | 217.00                                     | 5.2       | 229.30                                    | 0.2       |
| 5           | CQUAD4  | 247.90                 | 240.38                                     | 3.0       | 247.77                                    | 0.0       |
|             | CQUADR  | 247.90                 | 240.61                                     | 2.9       | 247.97                                    | 0.0       |
| 6           | CQUAD4  | 260.61                 | 252.52                                     | 3.1       | 259.98                                    | 0.2       |
|             | CQUADR  | 260.61                 | 252.85                                     | 3.0       | 260.33                                    | 0.1       |

### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 31.

## 5.2.12 Cantilevered Tapered Membrane

### Problem Description

Figure 1 shows the tapered membrane. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the tapered membrane are determined. All dimensions are in meters.

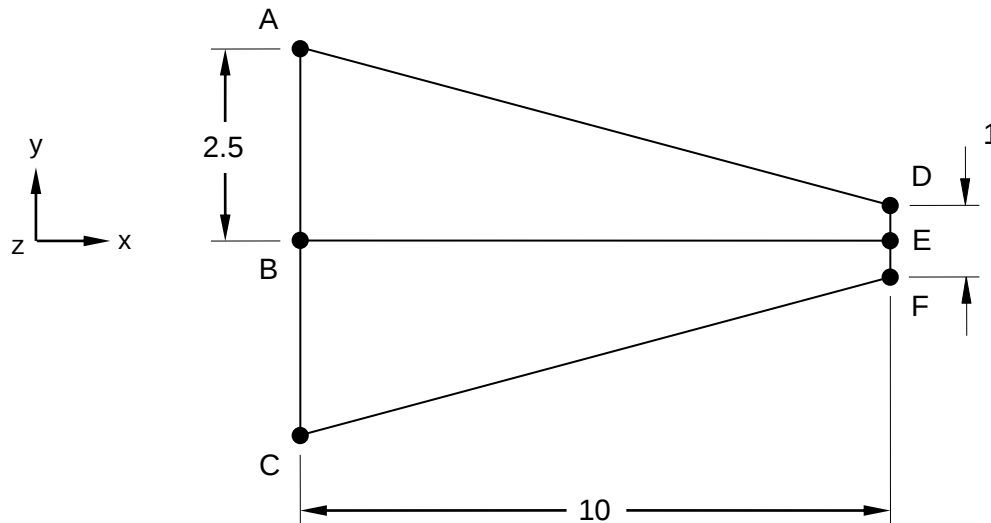


Figure 1. Cantilevered Tapered Membrane

### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_12c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm5\_2\_12d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm5\_2\_12cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation
- vm5\_2\_12dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation

### Model Data

#### Finite Element Modeling

- Attributes: Shear behavior. Irregular mesh. Symmetry.
- Test 1 and 2 (vm5\_2\_12c4 and vm5\_2\_12d4): 153 nodes, 128 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_12cR and vm5\_2\_12dR): 153 nodes, 128 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 4 elements on edges AB, BC, DE and EF, and 16 elements on edges AD, BE and CF (see Figure 1).

#### Units

meter/Newton/second

### Model Geometry

Length:  $L = 10.0$  m

Width:  $w_1 = 1$  m and  $w_2 = 5$  m

Thickness:  $t = 0.1$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Because the model is symmetric, only half of it is analyzed (defined by the points B, C, E, and F, as shown in Figure 1). The nodes along the Y-axis are fully constrained in all translations and rotations. All other nodes are constrained in all rotations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | Ref. Value             | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | CQUAD4  | 44.62                  | 44.56                                      | 0.1       | 44.65                                     | 0.1       |
|             | CQUADR  | 44.62                  | 44.52                                      | 0.2       | 44.64                                     | 0.1       |
| 2           | CQUAD4  | 130.03                 | 129.68                                     | 0.3       | 131.04                                    | 0.8       |
|             | CQUADR  | 130.03                 | 129.27                                     | 0.6       | 130.32                                    | 0.2       |
| 3           | CQUAD4  | 162.70                 | 162.61                                     | 0.1       | 162.80                                    | 0.0       |
|             | CQUADR  | 162.70                 | 162.62                                     | 0.1       | 162.76                                    | 0.0       |
| 4           | CQUAD4  | 246.05                 | 244.41                                     | 0.7       | 250.35                                    | 1.7       |
|             | CQUADR  | 246.05                 | 243.31                                     | 1.1       | 247.24                                    | 0.5       |
| 5           | CQUAD4  | 379.90                 | 374.87                                     | 1.3       | 391.59                                    | 3.1       |
|             | CQUADR  | 379.90                 | 372.85                                     | 1.9       | 383.00                                    | 0.8       |
| 6           | CQUAD4  | 391.44                 | 389.80                                     | 0.4       | 393.10                                    | 0.4       |
|             | CQUADR  | 391.44                 | 389.42                                     | 0.5       | 392.94                                    | 0.4       |

**Note:** Reference value (Ref. Value) refers to the accepted solution to this problem.

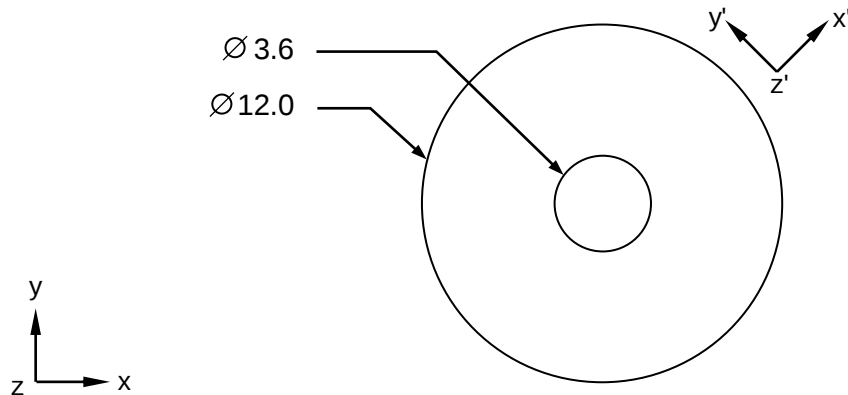
## References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 32.

### 5.2.13 Free Annular Membrane

#### Problem Description

Figure 1 shows the annular membrane. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the annular membrane are determined. All dimensions are in meters.



**Figure 1. Free Annular Membrane**

#### Autodesk Nastran Analysis Model Filenames

- vm5\_2\_13c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm5\_2\_13d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm5\_2\_13cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation
- vm5\_2\_13dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation

#### Model Data

##### *Finite Element Modeling*

- Attributes: Repeated eigenvalues. Rigid body modes (three modes). Kinematically incomplete suppression.
- Test 1 and 2 (vm5\_2\_13c4 and vm5\_2\_13d4): 192 nodes, 160 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5\_2\_13cR and vm5\_2\_13dR): 192 nodes, 160 6-DOF/node quadrilateral plate elements.
- The model has a mapped mesh with 32 elements on the inner and outer circles of the plate, respectively.

#### *Units*

meter/Newton/second

### Model Geometry

Outer Radius:  $R_o = 6.0$  m

Inner Radius:  $R_i = 1.8$  m

Thickness:  $t = 0.06$  m

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Nodes 254 and 286 (points A and B as shown in Figure 1) are constrained in the X and Y-translations. All other nodes are constrained in the Z-translation and all rotations.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 4, 5        | CQUAD4  | 129.51                 | 128.85                                     | 0.5       | 129.83                                    | 0.2       |
|             | CQUADR  | 129.51                 | 129.05                                     | 0.4       | 130.03                                    | 0.4       |
| 6           | CQUAD4  | 225.46                 | 224.52                                     | 0.4       | 225.22                                    | 0.1       |
|             | CQUADR  | 225.46                 | 224.55                                     | 0.4       | 225.25                                    | 0.1       |
| 7, 8        | CQUAD4  | 234.92                 | 230.88                                     | 1.7       | 236.16                                    | 0.5       |
|             | CQUADR  | 234.92                 | 230.90                                     | 1.7       | 236.17                                    | 0.5       |
| 9, 10       | CQUAD4  | 272.13                 | 265.21                                     | 2.5       | 272.17                                    | 0.0       |
|             | CQUADR  | 272.13                 | 265.69                                     | 2.4       | 272.62                                    | 0.2       |
| 11, 12      | CQUAD4  | 340.34                 | 329.45                                     | 3.2       | 341.00                                    | 0.2       |
|             | CQUADR  | 340.34                 | 329.52                                     | 3.2       | 341.06                                    | 0.2       |
| 13, 14      | CQUAD4  | 391.98                 | 369.76                                     | 5.7       | 390.95                                    | 0.3       |
|             | CQUADR  | 391.98                 | 371.12                                     | 5.7       | 392.16                                    | 0.0       |

### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 33.

### 5.3 Axisymmetric Solid and Solid Element Test Cases

The following normal mode/eigenvalue verification problems using standard NAFEMS Benchmarks are performed using solid elements

#### 5.3.1 Simply – Supported “Solid” Square Plate

##### Problem Description

Figure 1 shows the simply supported square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method for both lumped mass and consistent mass. The natural frequencies of the simply supported square plate are determined. All dimensions are in meters.

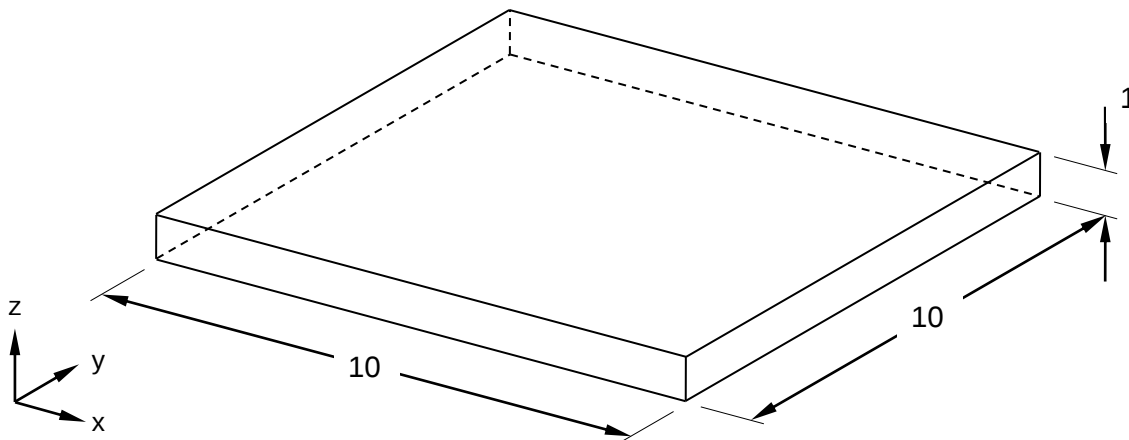


Figure 1. Simply – Supported “Solid” Square Plate

##### Autodesk Nastran Analysis Model Filenames

- vm5\_3\_1cl.nas – Linear solid brick elements, coupled mass formulation
- vm5\_3\_1dl.nas – Linear solid brick elements, diagonal mass formulation
- vm5\_3\_1cp.nas – Parabolic solid brick elements, coupled mass formulation
- vm5\_3\_1dp.nas – Parabolic solid brick elements, diagonal mass formulation

##### Model Data

##### *Finite Element Modeling*

- Attributes: Well established. Rigid body modes (three modes). Kinematically incomplete suppressions.
- Tests 1 and 2 (vm5\_3\_1cl, vm5\_3\_1dl): 324 nodes, 192 5 Linear solid brick elements.
- Tests 3 and 4 (vm5\_3\_1cp, vm5\_3\_1dp): 155 nodes, 16 parabolic solid brick elements.

##### *Units*

meter/Newton/second

**Model Geometry**

Length:  $L = 10.0$  m

Width:  $w = 10.0$  m

Thickness:  $t = 1$  m

**Material Properties**

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

Constraint set 1: All the nodes along the four edges on the XY plane at  $Z = -0.5$  m are constrained in the Z-translation.

Constraint set 2 (Kinematic DOF): Tests 1 and 2: Nodes 36 and 264 are constrained in all translations. Tests 3 and 4: Nodes 27 and 219 are constrained in the X and Y-translations.

**Solution Type**

Normal Modes/Eigenvalue – Subspace iterative method

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | Element   | Ref. Value             | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|-----------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |           | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 4           | Linear    | 45.90                  | 44.11                                      | 3.9       | 45.32                                     | 1.3       |
|             | Parabolic | 45.90                  | 44.47                                      | 3.1       | 44.78                                     | 2.4       |
| 5, 6        | Linear    | 109.44                 | 106.72                                     | 2.5       | 113.96                                    | 4.1       |
|             | Parabolic | 109.44                 | 107.81                                     | 1.5       | 107.56                                    | 1.7       |
| 7           | Linear    | 167.89                 | 156.47                                     | 6.8       | 173.30                                    | 3.2       |
|             | Parabolic | 167.89                 | 161.13                                     | 4.0       | 169.90                                    | 1.2       |
| 8           | Linear    | 193.59                 | 193.58                                     | 0.0       | 196.77                                    | 1.6       |
|             | Parabolic | 193.59                 | 185.54                                     | 4.2       | 192.74                                    | 0.4       |
| 9, 10       | Linear    | 206.19                 | 200.14                                     | 2.9       | 209.56                                    | 1.6       |
|             | Parabolic | 206.19                 | 192.39                                     | 6.7       | 205.57                                    | 0.3       |

**Note:** Reference value (Ref. Value) refers to the accepted solution to this problem.



## References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 52.

## 6. Verification Test Cases from the Societe Francaise des Mechaniciens

The purpose of these test cases is to verify the functionality of Autodesk Nastran using standard benchmarks published by SFM (Societe Francaise des Mecaniciens, Paris, France) in “Guide de validation des progiciels de calcul de structures”.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent common and well-known applications of the finite element method.

## 6.1 Mechanical Structures – Linear Statics Analysis with Bar or Rod Elements

The following mechanical structures verification problems using standard SFM Benchmarks are performed using linear static analysis with bar or rod elements.

### 6.1.1 Short Beam on Two Articulated Supports

#### Problem Description

Figure 1 shows the short beam with two fixed supports. Static analysis is performed on the model. The total translation at point B (node 7) of the beam is determined. All dimensions are in millimeters.

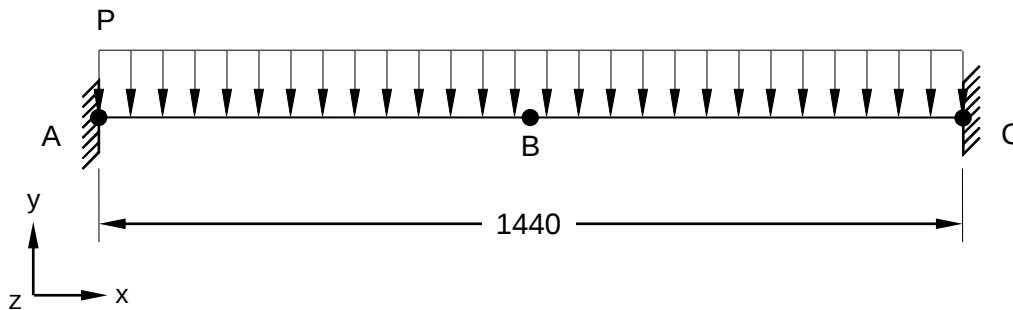


Figure 1. Short Beam on Two Articulated Supports

#### Autodesk Nastran Analysis Model Filenames

- vm6\_1\_1.nas

#### Model Data

##### *Finite Element Modeling*

- 11 nodes, 10 bar elements

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L = 1.440$  m

##### *Cross Sectional Properties*

Area:  $A = 31.0 \text{ E-4 m}^2$

Moment of Inertia:  $I = 2810.0 \text{ E-8 m}^4$

Shear Area Ratio = 2.42

**Material Properties**

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

The two nodes at the ends of the beam (nodes 1 and 2, at points A and C, as shown in Figure 1) are constrained in all translations and rotations except for the Z-rotation. A load  $P = 1.0 \text{ E}+5 \text{ N/m}$  is applied to nodes 1-10 in the negative Y-direction.

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                               | Theory     | Autodesk Nastran | Error (%) |
|-------------------------------------------|------------|------------------|-----------|
| Total Translation at Point B (node 7) (m) | -1.2593E-2 | -1.2491E-2       | 0.8       |

**References**

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLL02/89.

## 6.1.2 Clamped Beams Linked by a Rigid Element

### Problem Description

Figure 1 shows the clamped beams linked by a rigid element. Static analysis is performed on the model. The displacement, constraint moment, and constraint force on the beams are determined. All dimensions are in millimeters.

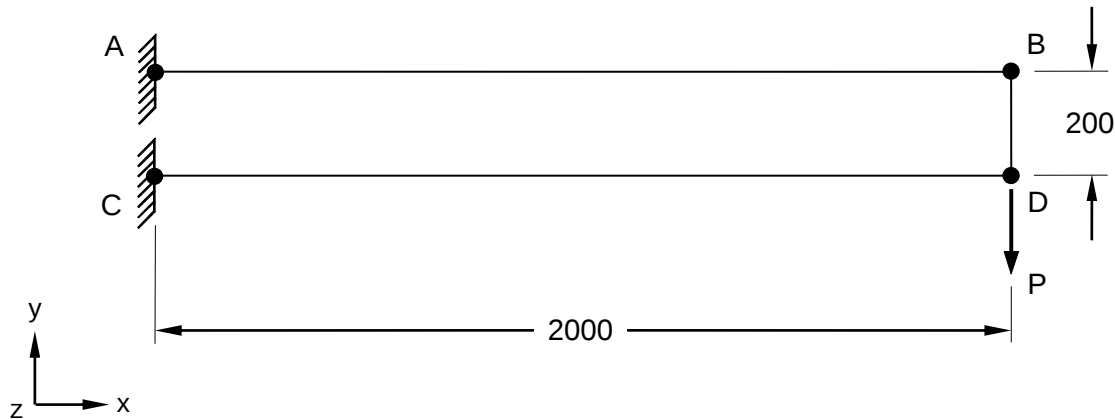


Figure 1. Clamped Beams Linked by a Rigid Element

### Autodesk Nastran Analysis Model Filenames

- vm6\_1\_2.nas

### Model Data

#### *Finite Element Modeling*

- 22 nodes, 20 bar elements (10 on each edge, AB and CD, respectively), and 1 rigid element (on edge BD).

### Units

meter/Newton/second

### Model Geometry

Length:  $L = 2$  m

Width:  $w = 0.2$  m

### Cross Sectional Properties

Moment of Inertia:  $I = 1.33 \text{ E-}8 \text{ m}^4$

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E+}9 \text{ N/m}^2$

### Boundary Conditions

Points A (node 1) and C (node 4) are fully constrained in all translations and rotations. A nodal force  $P = 1000 \text{ N}$  is set to point D (node 3) in the negative Y-direction.

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                             | Node | Point | Theory | Autodesk Nastran | Error (%) |
|-----------------------------------------|------|-------|--------|------------------|-----------|
| Displacement Y (T2 Translation) (m)     | 6    | B     | -0.125 | -0.125           | 0.0       |
| Displacement Y (T2 Translation) (m)     | 3    | D     | -0.125 | -0.125           | 0.0       |
| Force Y (T2 Constraint Force) (N)       | 1    | A     | 500    | 500              | 0.0       |
| Moment $R_z$ (R3 Constraint Force) (Nm) | 1    | A     | 500    | 500              | 0.0       |
| Force Y (T2 Constraint Force) (N)       | 4    | C     | 500    | 500              | 0.0       |
| Moment $R_z$ (R3 Constraint Force) (Nm) | 4    | C     | 500    | 500              | 0.0       |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLL05/89.

### 6.1.3 Plane Bending Load on a Thin Arc

#### Problem Description

Figure 1 shows the model of the thin arc and applied force. Static analysis is performed on the model. The displacements of the thin arc are determined.

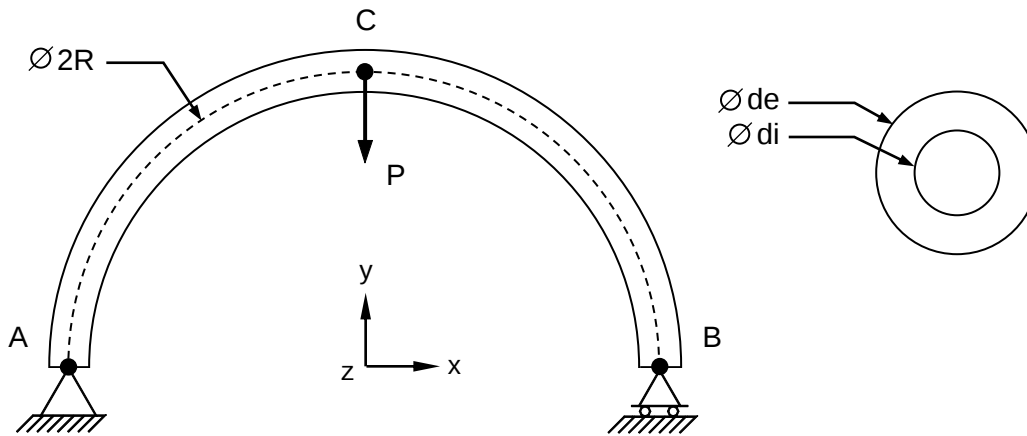


Figure 1. Plane Bending Load on a Thin Arc

#### Autodesk Nastran Analysis Model Filenames

- vm6\_1\_3.nas

#### Model Data

##### *Finite Element Modeling*

- 11 nodes, 10 bar elements

##### *Units*

meter/Newton/second

##### *Model Geometry*

Radius of Arc:  $R = 1$  m

External Diameter of Bar:  $d_e = 0.020$  m

Inside Diameter of Bar:  $d_i = 0.016$  m

### Cross Sectional Properties

Area:  $A = 1.131 \text{ E-4 m}^2$

Moment of Inertia:  $I_x = 4.637 \text{ E-9 m}^4$

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E+9 N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Point A (node 2) is constrained in all translations. Point B (node 1) is constrained in the Y and Z-translations. A nodal force  $P = 100 \text{ N}$  is set to point C in the negative Y-direction.

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description               | Node | Point | Theory     | Autodesk Nastran | Error (%) |
|---------------------------|------|-------|------------|------------------|-----------|
| $R_z$ (R3 Rotation) (rad) | 2    | A     | -3.0774E-2 | -3.1097E-2       | 1.0       |
| $R_z$ (R3 Rotation) (rad) | 1    | B     | 3.0774E-2  | 3.1097E-2        | 1.0       |
| Y (T2 Translation) (m)    | 7    | C     | -1.9206E-2 | -1.9342E-2       | 0.7       |
| X (T1 Translation) (m)    | 1    | B     | 5.3913E-2  | 5.3734E-2        | 0.3       |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLL08/89.



### 6.1.4 Articulated Plane Truss

#### Problem Description

Figure 1 shows the articulated plane truss and applied forces. Static analysis is performed on the model. The displacement, and reaction force of the plane truss are determined. All dimensions are in meters.

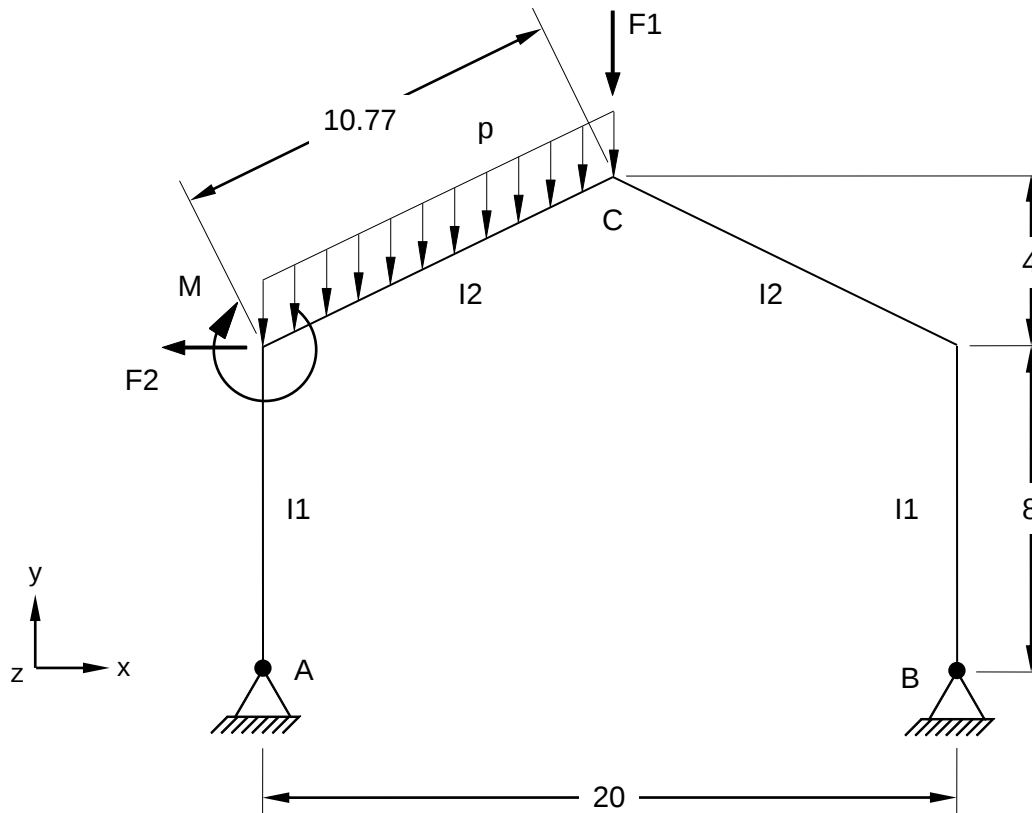


Figure 1. Articulated Plane Truss

#### Autodesk Nastran Analysis Model Filenames

- vm6\_1\_4a.nas – 4 bar elements
- vm6\_1\_4b.nas – 10 bar elements

#### Model Data

##### Finite Element Modeling

- Test 1 (vm6\_1\_4a): 5 nodes, 4 bar elements (one on each of the four parts of the truss).
- Test 2 (vm6\_1\_4b): 11 nodes, 10 bar elements (2 on each vertical part of the truss and 3 on each inclined part of the truss).

#### Units

meter/Newton/second

### Model Geometry

Length:  $L = 20$  m

Height:  $h_1 = 8$  m and  $h_2 = 4$  m

Inside Diameter of Bar:  $d_i = 0.016$  m

### Cross Sectional Properties

Moment of Inertia:  $I_1 = 5.0 \text{ E-4 m}^4$

Moment of Inertia:  $I_2 = 2.5 \text{ E-4 m}^4$

### Material Properties

Young's Modulus:  $E = 210.0 \text{ E+9 N/m}^2$

### Boundary Conditions

Test 1 (vm6\_1\_4a): Points A (node 1) and B (node 4) are constrained in all translations. Nodes 2, 3, and 8 are constrained in the Z-translation. The forces and moments are set to the following numerical values:  $p = -3,000 \text{ N/m}$  (on element 4);  $F_1 = -20,000 \text{ N}$  (on node 8);  $F_2 = -10,000 \text{ N}$  (on node 2), and  $M = -100,000 \text{ Nm}$  (on node 2).

Test 2 (vm6\_1\_4b): Points A (node 1) and B (node 4) are constrained in all translations. Nodes 2, 3, 5-13 are constrained in the Z-translation. The forces and moments are set to the following numerical values:  $p = -3,000 \text{ N/m}$  (on element 5-7);  $F_1 = -20,000 \text{ N}$  (on node 8);  $F_2 = -10,000 \text{ N}$  (on node 2), and  $M = -100,000 \text{ Nm}$  (on node 2).

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                       | Node | Point | Theory    | Test | Autodesk Nastran | Error (%) |
|---------------------------------------------------|------|-------|-----------|------|------------------|-----------|
| Vertical (Y) reaction (T2 Constraint force) (N)   | 2    | A     | 31,500    | 1    | 33,233           | 5.5       |
|                                                   |      |       |           | 2    | 33,233           | 5.5       |
| Horizontal (X) reaction (T1 Constraint force) (N) | 1    | B     | 20,239    | 1    | 20,609           | 1.8       |
|                                                   |      |       |           | 2    | 20,609           | 1.8       |
| Y (T2 Translation) (m)                            | 7    | C     | -3.072E-2 | 1    | -3.1609E-2       | 2.9       |
|                                                   |      |       |           | 2    | -3.1609E-2       | 2.9       |

**Note:** The theoretical values neglect the shear effect.

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLL14/89.

## 6.2 Mechanical Structures – Linear Statics Analysis with Plate Elements

The following mechanical structures verification problems using standard SFM Benchmarks are performed using linear static analysis with plate elements.

### 6.2.1 Plane Shear and Bending Load on a Plate

#### Problem Description

Figure 1 shows the plate and applied plane shear load. Static analysis is performed on the model. The centerline displacement of the plate is determined. All dimensions are in millimeters.

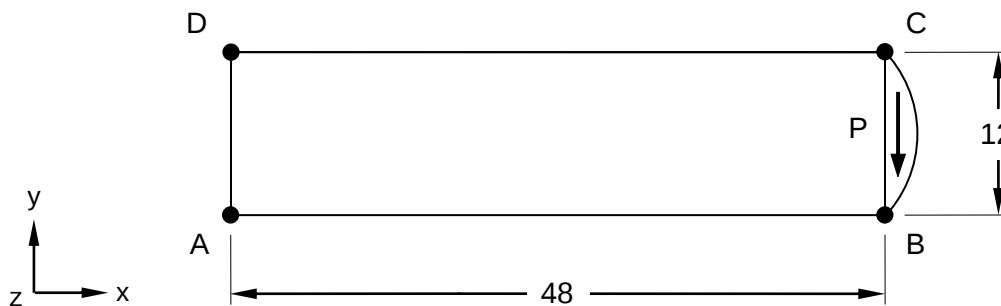


Figure 1. Plane Shear and Bending Load on a Plate

#### Autodesk Nastran Analysis Model Filenames

- vm6\_2\_1.nas

#### Model Data

##### *Finite Element Modeling*

- 126 nodes, 100 5-DOF/node quadrilateral plate elements (5 elements on AD and BC edges, and 20 elements on AB and CD edges, as shown in Figure 1).

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 0.048$  m

Width:  $w = 0.012$  m

Thickness:  $t = 0.001$  m

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.25$

**Boundary Conditions**

Nodes 121-126 (on the AD edge) are fully constrained in all translations and rotations. A shear force with parabolic distribution on the width and constant distribution on the thickness is set on the BC edge. The resultant force (P) is 40 N.

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                        | Point Coordinates | Node | Theory | Autodesk Nastran | Error (%) |
|------------------------------------|-------------------|------|--------|------------------|-----------|
| Y centerline (T2 Translation) (mm) | (L,y)             | 3    | 0.3413 | 0.3408           | 0.15      |

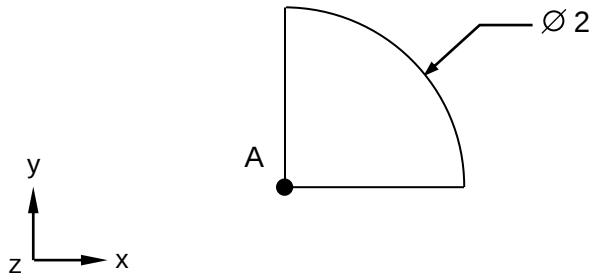
**References**

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLP01/89.

## 6.2.2 Uniformly Distributed Load on a Circular Plate

### Problem Description

Figure 1 shows the circular plate ( $\frac{1}{4}$  of it). Static analysis is performed on the model. The translation displacement for both test cases is determined. All dimensions are in meters.



**Figure 1. Uniformly Distributed Load on a Circular Plate**

### Autodesk Nastran Analysis Model Filenames

- vm6\_2\_2a.nas – 5-DOF/node quadrilateral elements
- vm6\_2\_2b.nas – 5-DOF/node triangular elements

### Model Data

#### *Finite Element Modeling*

- Test 1 (vm6\_2\_2a): Free meshing 166 nodes, 144 5-DOF/node quadrilateral plate elements.
- Test 2 (vm6\_2\_2b): Free meshing 149 nodes, 254 5-DOF/node triangular plate elements.
- Only  $\frac{1}{4}$  of the plate is meshed.

#### *Units*

meter/Newton/second

#### *Model Geometry*

Radius:  $R = 1$  m

Thickness:  $t = 0.005$  m

#### *Material Properties*

Young's Modulus:  $E = 210.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

Node 1 (at point A) is constrained in all translations and rotations except the Z-translation. The nodes along the arc are fully constrained in all translations and rotations. The nodes along the X and Y-axis are constrained using symmetry boundary conditions. A uniform elemental pressure of  $p = -1,000$  Pa is applied.

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                         | Test | Node | Point    | Theory  | Autodesk Nastran | Error (%) |
|-------------------------------------|------|------|----------|---------|------------------|-----------|
| T3 Translation (Displacement Z) (m) | 1    | 1    | Center 0 | -6.5E-3 | -4.2E-3          | 35.4      |
| T3 Translation (Displacement Z) (m) | 2    | 1    | Center 0 | -6.5E-3 | -5.8E-3          | 10.8      |

**References**

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS03/89.

### 6.2.3 Torque Loading on a Square Tube

#### Problem Description

Figure 1 shows the square tube (0.1 by 0.1 m, and 5 mm thickness) with torque loading. Static analysis is performed on the model. The displacement and stress on the square tube is determined. All dimensions are in meters.

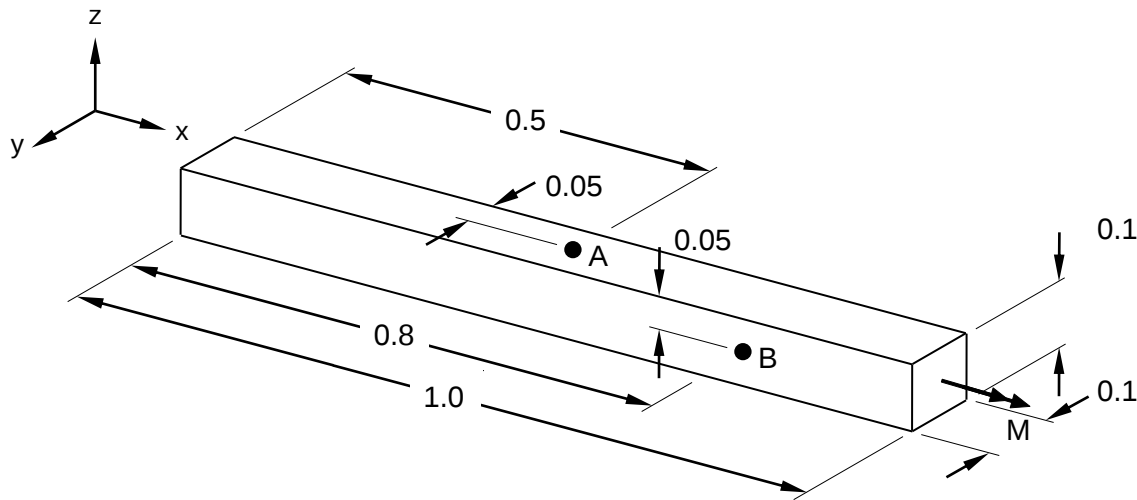


Figure 1. Torque Loading on a Square Tube

#### Autodesk Nastran Analysis Model Filenames

- vm6\_2\_3.nas

#### Model Data

##### Finite Element Modeling

- Mapped meshing 176 nodes, 160 5-DOF/node quadrilateral plate elements (4 elements on the small edges and 10 elements on the long edges of the tube).

#### Units

meter/Newton/second

#### Model Geometry

Length:  $L = 1.0$  m

Width:  $w = 0.1$  m

Height:  $h = 0.1$  m

Thickness:  $t = 0.005$  m

#### Material Properties

Young's Modulus:  $E = 210.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Nodes 1-5, 112-115, and 167-169 (on the left end of the tube) are constrained in all translations and rotations. A torque  $M = 10 \text{ N-m}$  is applied to the free end of the tube (on the right hand side of the tube).

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                    | Node | Point | Theory   | Autodesk Nastran | Error (%) |
|--------------------------------|------|-------|----------|------------------|-----------|
| T2 Translation (m)             | 193  | A     | -6.2E-7  | -6.2E-7          | 0.0       |
| R1 Rotation (rad)              | 193  | A     | 1.2E+5   | 1.2E+5           | 0.0       |
| Plate Bottom Minor Stress (Pa) | 193  | A     | -110,000 | -109,369         | 5.7       |
| T2 Translation (m)             | 208  | B     | -9.9E-7  | -9.9E-7          | 0.0       |
| R1 Rotation (rad)              | 208  | B     | 2.0E+5   | 2.0E+5           | 0.0       |
| Plate Bottom Minor Stress (Pa) | 208  | B     | -110,000 | -108,558         | 13.1      |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS05/89.



## 6.2.4 Cylindrical Shell with Internal Pressure

### Problem Description

Figure 1 shows the model of the cylindrical shell (1/8 model, with thickness = 20 mm). Static analysis is performed. The displacements and the stresses of the cylindrical shell with an internal pressure are determined. All dimensions are in meters.

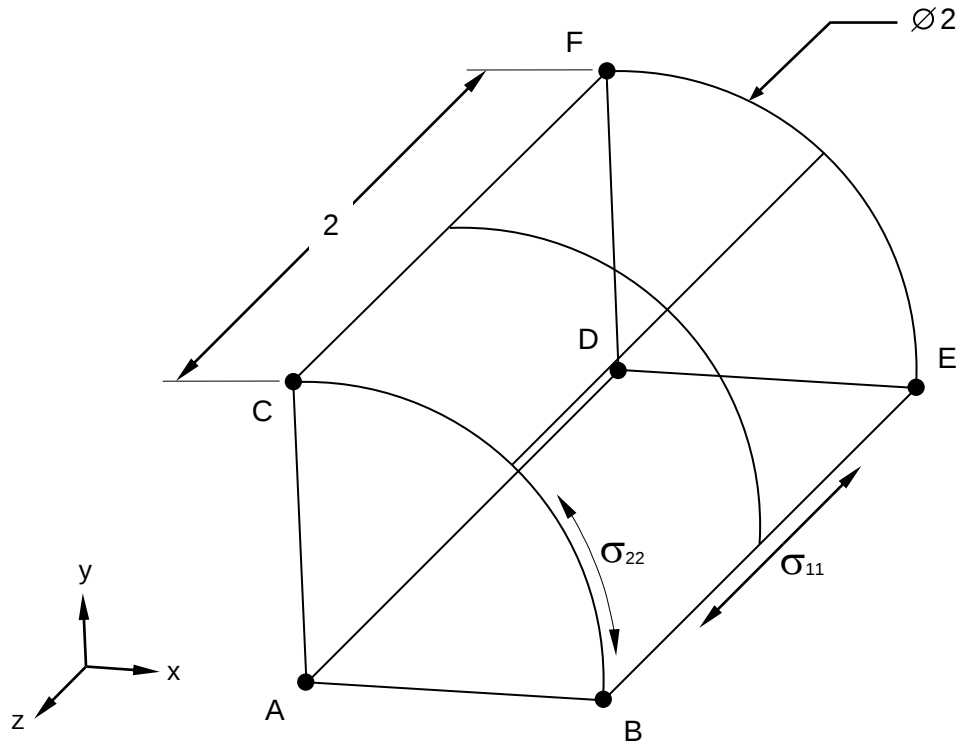


Figure 1. Cylindrical Shell with Internal Pressure

### Autodesk Nastran Analysis Model Filenames

- vm6\_2\_4a.nas – 6-DOF/node quadrilateral elements
- vm6\_2\_4b.nas – 6-DOF/node quadrilateral elements

### Model Data

#### Finite Element Modeling

- Test 1 (vm6\_2\_4a): 121 nodes, 100 6-DOF/node quadrilateral plate elements (10 elements on each of the following edges: BC, EF, BE and CF).

- Test 2 (vm6\_2\_4b): 441 nodes, 400 6-DOF/node quadrilateral plate elements (20 elements on each of the following edges: BC, EF, BE and CF).

### Units

meter/Newton/second

### Model Geometry

Length:  $L = 2.0$  m

Radius:  $R = 1.0$  m

Thickness:  $t = 0.02$  m

### Material Properties

Young's Modulus:  $E = 210.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

The model is constrained using symmetry boundary conditions. One end of the model is constrained as a cap covering the end of the cylinder. The opposite side (edge BC) is constrained in X and Y-rotations because the CQUADR element needs these additional boundary conditions due to the hidden mid-side nodes. An internal elemental pressure of 10,000 Pa is applied to the model (from inside out).

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                  | Test | Node | Theory   | Autodesk Nastran | Error (%) |
|----------------------------------------------|------|------|----------|------------------|-----------|
| $\sigma_{11}$ Plate Top Y Normal Stress (Pa) | 1    | 11   | 0.0      | 1.48             | NA        |
|                                              | 2    | 21   | 0.0      | 0.63             | NA        |
| $\sigma_{22}$ Plate Top X Normal Stress (Pa) | 1    | 111  | 500,000  | 498,464          | 0.3       |
|                                              | 2    | 421  | 500,000  | 499,617          | 0.1       |
| $\Delta R_{T1}$ Translation (m)              | 1    | 121  | 2.38E-6  | 2.37E-6          | 0.4       |
|                                              | 2    | 441  | 2.38E-6  | 2.38E-6          | 0.0       |
| $\Delta R_{T3}$ Translation (m)              | 1    | 121  | -1.43E-6 | -1.42E-6         | 0.7       |
|                                              | 2    | 441  | -1.43E-6 | -1.43E-6         | 0.0       |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS06/89.

## 6.2.5 Gravity Loading on a Thin Wall Cylinder

### Problem Description

Figure 1 shows the model of the thin wall cylinder ( $\frac{1}{4}$  of the model, with thickness = 20 mm). Static analysis is performed. The displacements and stresses of the thin wall cylinder with gravity loading are determined. All dimensions are in meters.

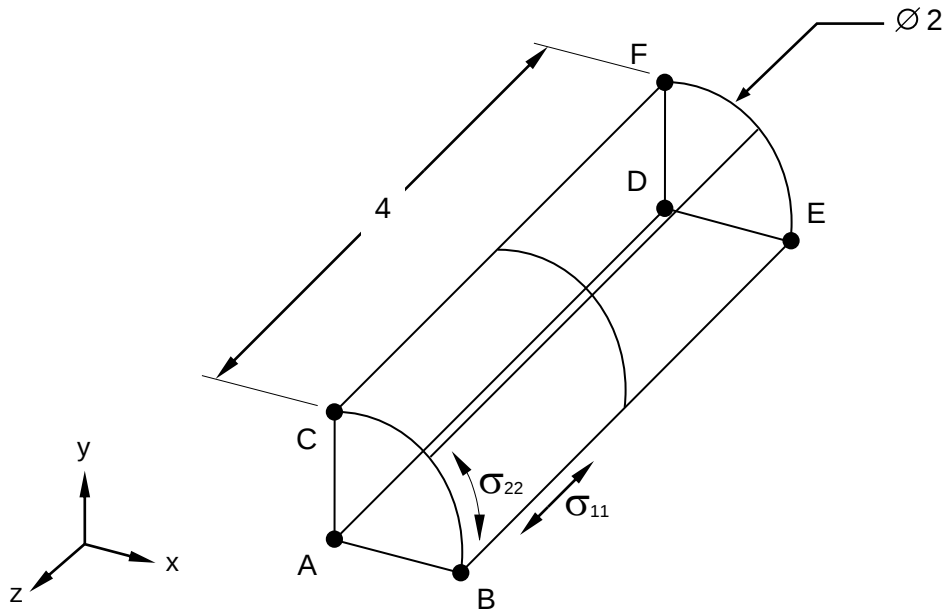


Figure 1. Gravity loading on a Thin Wall Cylinder

### Autodesk Nastran Analysis Model Filenames

- vm6\_2\_5.nas

### Model Data

#### Finite Element Modeling

- Mapped Meshing: 84 nodes, 65 5-DOF/node quadrilateral plate elements.

### Units

meter/Newton/second

### Model Geometry

Length:  $L = 4.0$  m

Radius:  $R = 1.0$  m

Thickness:  $t = 0.02$  m

### Material Properties

Young's Modulus:  $E = 210.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

Specific Weight:  $\gamma = 7.85 \text{ E}+4 \text{ N/m}^3$

Mass Density:  $\rho = 8002.0 \text{ kg/m}^3$

### Boundary Conditions

Symmetry is applied to the two radial edges (BE and CF). The top edge (EF) is constrained in the Z-translation only. A translational acceleration body load of  $9.8 \text{ m/s}^2$  is applied in the negative Z-direction.

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                  | Node | Point | Theory   | Autodesk Nastran | Error (%) |
|----------------------------------------------|------|-------|----------|------------------|-----------|
| $\sigma_{11}$ Plate Top X Normal Stress (Pa) | 2    | X=0   | 314,000  | 302,227          | 3.7       |
| $\sigma_{22}$ Plate Top Y Normal Stress (Pa) | 1    | Any   | 0.0      | 0.0              | 0.0       |
| $\Delta_R$ T1 Translation (m)                | 2    | X=0   | -4.49E-7 | -4.39E-7         | 2.0       |
| $\Delta_z$ T3 Translation (m)                | 1    | X=L   | 2.99E-6  | 2.99E-6          | 0.0       |
| $\Psi$ R2 Rotation (rad)                     | 10   | X-L   | -1.12E-7 | -1.12E-7         | 0.0       |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS09/89.

## 6.2.6 Pinched Cylindrical Shell

### Problem Description

Figure 1 shows the cylindrical shell (1/4 of the model, with thickness = 0.094 mm). Static analysis is performed. The displacements of the cylindrical shell are determined. All dimensions are in meters.

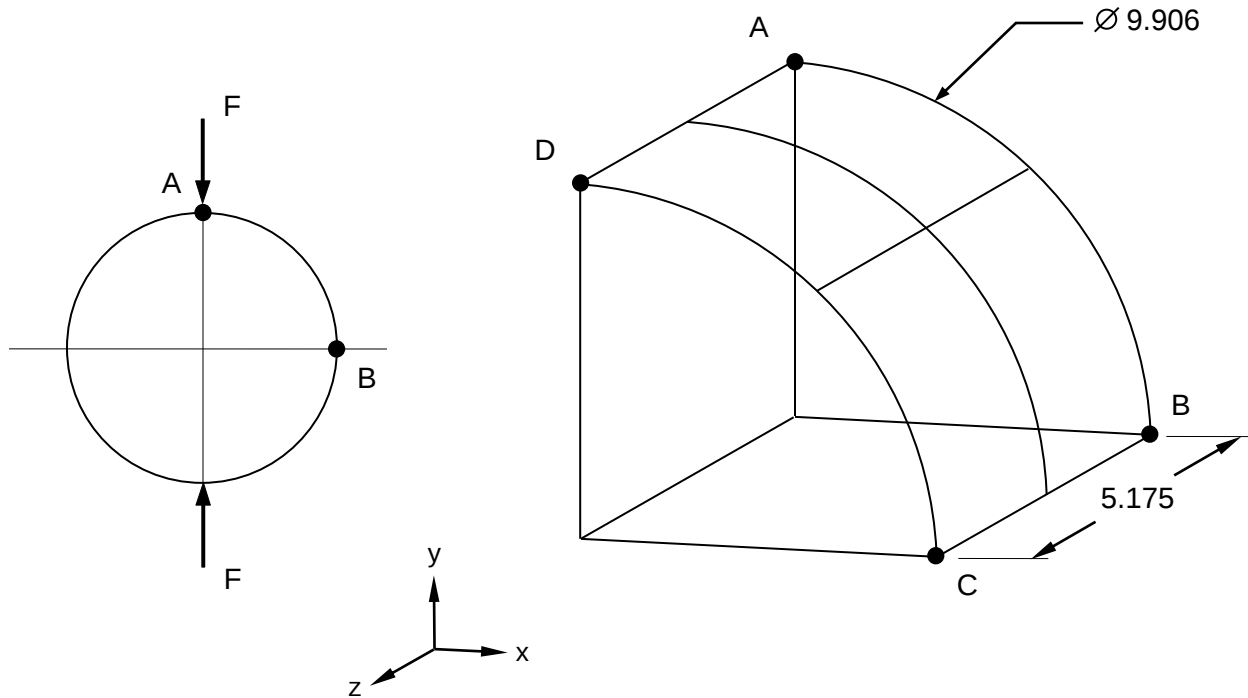


Figure 1. Pinched Cylindrical Shell

### Autodesk Nastran Analysis Model Filenames

- vm6\_2\_6a.nas – 5-DOF/node triangular plate elements
- vm6\_2\_6b.nas – 5-DOF/node quadrilateral plate elements

### Model Data

#### *Finite Element Modeling*

- Test 1 (vm6\_2\_6a): Free meshing. 173 nodes, 296 5-DOF/node triangular plate elements.
- Test 2 (vm6\_2\_6b): Mapped meshing. 165 nodes, 140 5-DOF/node quadrilateral plate elements.

**Units**

meter/Newton/second

**Model Geometry**Length:  $L = 5.175$  mRadius:  $R = 4.953$  mThickness:  $t = 0.094$  m**Material Properties**Young's Modulus:  $E = 10.5 \text{ E}+6 \text{ N/m}^2$ Poisson's Ratio:  $\nu = 0.3125$ **Boundary Conditions**

Symmetry is applied about the XY, XZ and YZ-planes. Point C is fully constrained in all translations and rotations, except the X-translation. Point B is constrained in the Y-translation, and X and Z-rotations. Point D is fully constrained in all translations and rotations except the Y-translation. Point A is constrained in the X-translation, and Y and Z-rotations. Edge CD is constrained in the Z-translation, and X and Y-rotations. Nodal forces  $F = 25$  N are applied at point D (in the negative Y-direction).

**Solution Type**

Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                  | Test | Point | Theory  | Autodesk Nastran | Error (%) |
|----------------------------------------------|------|-------|---------|------------------|-----------|
| Displacement Y (Node 3) (T2 Translation) (m) | 1    | D     | -0.1139 | -0.1067          | 6.3       |
| Displacement Y (Node 3) (T2 Translation) (m) | 2    | D     | -0.1139 | -0.1126          | 1.1       |

**References**

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS20/89.

## 6.2.7 Infinite Plate with a Circular Hole

### Problem Description

Figure 1 shows  $\frac{1}{4}$  of the plate model, with thickness = 0.001 m. Static analysis is performed. The plate top normal stress at points A, B and C are determined. All dimensions are in meters.

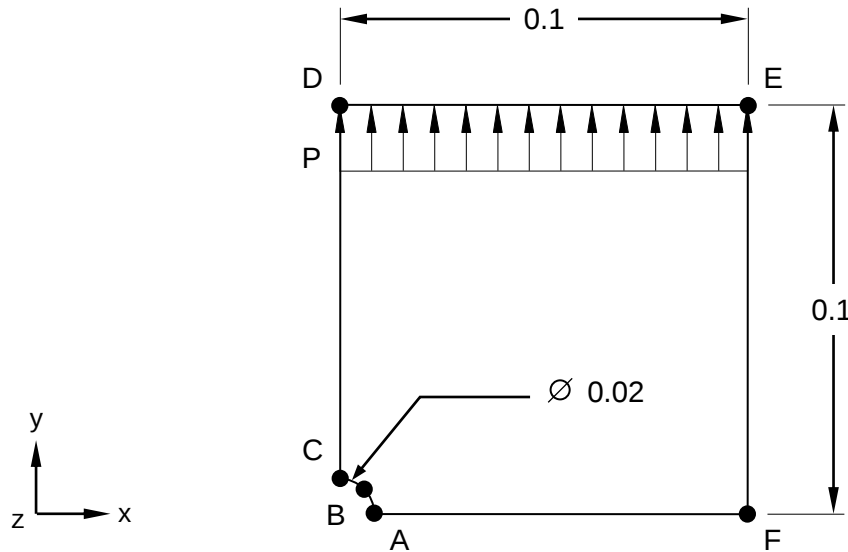


Figure 1. Infinite Plate with a Circular Hole

### Autodesk Nastran Analysis Model Filenames

- vm6\_2\_7

### Model Data

#### Finite Element Modeling

- A  $\frac{1}{4}$  model is created using 121 nodes, and 100 5-DOF/node quadrilateral plate elements.

#### Units

meter/Newton/second

#### Model Geometry

Length:  $L = 0.1$  m

Radius:  $R = 0.01$  m

Thickness:  $t = 0.001$  m

#### Material Properties

Young's Modulus:  $E = 3.0 \text{ E}+10$  Pa

Poisson's Ratio:  $\nu = 0.25$

### Boundary Conditions

The model is constrained using symmetry boundary conditions (on edges AF and CD, as shown in Figure 1). Edge AF is constrained in the Y and Z-translations, and X and Z-rotations. Edge CD is constrained in the X and Z-translations, and Y and Z-rotations. All other nodes are constrained in the Z-translation only. An edge force  $P = 2500 \text{ N/m}$  in the negative Y-direction is applied on edge DE. This represents a uniform pressure on the edge of  $2.5 \text{ N/mm}^2$ .

### Solution Type

Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                        | Theory | Autodesk Nastran | Error (%) |
|------------------------------------|--------|------------------|-----------|
| Top Normal Stress at Point A (MPa) | 7.5    | 7.55             | 0.7       |
| Top Normal Stress at Point B (MPa) | 2.5    | 2.52             | 0.8       |
| Top Normal Stress at Point C (MPa) | -2.5   | -2.45            | 2.0       |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS02/89.



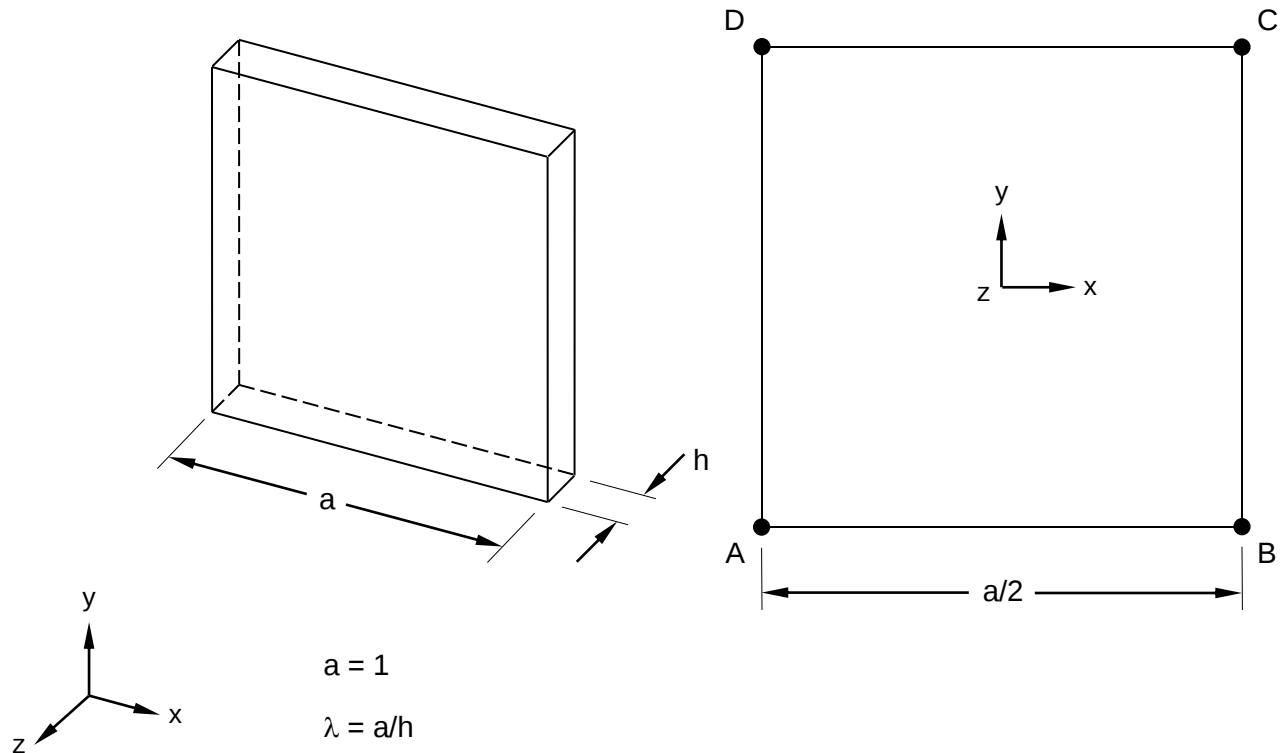
### 6.3 Mechanical Structures – Linear Statics Analysis with Solid Elements

The following mechanical structures verification problems using standard SFM Benchmarks are performed using linear static analysis with solid elements.

#### 6.3.1 Thick Plate Clamped at Edges

##### Problem Description

Figure 1 shows the thick plate (1/4 model is used with different thickness). Static analysis is performed. The translation at point C of the thick plate is determined. All dimensions are in meters.



**Figure 1. Thick Plate Clamped at Edges**

##### Autodesk Nastran Analysis Model Filenames

- vm6\_3\_1a10.nas – Test 1, 3-DOF/node brick,  $\lambda = 10$
- vm6\_3\_1a20.nas – Test 1, 3-DOF/node brick,  $\lambda = 20$
- vm6\_3\_1a50.nas – Test 1, 3-DOF/node brick,  $\lambda = 50$
- vm6\_3\_1a75.nas – Test 1, 3-DOF/node brick,  $\lambda = 75$
- vm6\_3\_1a100.nas – Test 1, 3-DOF/node brick,  $\lambda = 100$
- vm6\_3\_1b10.nas – Test 2, 5-DOF/node plate,  $\lambda = 10$
- vm6\_3\_1b20.nas – Test 2, 5-DOF/node plate,  $\lambda = 20$
- vm6\_3\_1b50.nas – Test 2, 5-DOF/node plate,  $\lambda = 50$
- vm6\_3\_1b75.nas – Test 2, 5-DOF/node plate,  $\lambda = 75$
- vm6\_3\_1b100.nas – Test 2, 5-DOF/node plate,  $\lambda = 100$

## Model Data

### *Finite Element Modeling*

- Test 1: Mapped meshing. 228 nodes, 25 3-DOF/node brick elements.  $\lambda = 10$ ,  $\lambda = 20$ ,  $\lambda = 50$ ,  $\lambda = 75$ ,  $\lambda = 100$ .
- Test 2: Mapped meshing. 36 nodes, 25 5-DOF/node quadrilateral plate elements.  $\lambda = 10$  &  $t=0.1$ ,  $\lambda = 20$  &  $t = 0.05$ ,  $\lambda = 50$  &  $t = 0.02$ ,  $\lambda = 75$  &  $t = 0.013$ ,  $\lambda = 100$  &  $t = 0.01$ .

### *Units*

meter/Newton/second

### *Material Properties*

*Young's Modulus:*  $E = 2.1 \text{ E}+11 \text{ N/m}^2$

*Poisson's Ratio:*  $\nu = 0.3$

### *Boundary Conditions*

Constraints Test 1: The nodes on edges AB, A'B', AD, and A'D' are constrained in all translations and rotations. The nodes on edge BC and B'C' are constrained in the X-translation, and Y and Z-rotations. The corner nodes at C and C' are constrained in all translations and rotations, except for the Z-translation. The nodes on edge DC and D'C' are constrained in the Y-translation, and X and Z-rotations.

Constraints Test 2: The nodes on edges AB and AD are fully constrained in all translations and rotations. The nodes on edge BC are constrained in the X-translation, and Y and Z-rotations. The corner nodes at C are constrained in all translations and rotations except for the Z-translation. The nodes on edge DC are constrained in the Y-translation, and X and Z-rotations.

Load case 1: An elemental pressure  $p = 1\text{E}+6 \text{ Pa}$  is applied in the negative Z-direction.

Load case 2: A nodal force is applied at point C of  $F = 1\text{E}+6 \text{ N}$  in the negative Z-direction.

### *Solution Type*

Static

## Comparison of Results

The tabular results are given in Tables 1 and 2.

**Table 1. Test Case 1 Results (T3 Translation at location C) (m)**

| Element | $\lambda^*$ | Load Case | Node | SFDM       | Autodesk Nastran | Error (%) |
|---------|-------------|-----------|------|------------|------------------|-----------|
| CHEXA   | 10          | Pressure  | 242  | -7.623E-5  | -7.443E-5        | 2.4       |
| CHEXA   | 10          | Force     | 242  | -4.2995E-4 | -4.3582E-4       | 1.4       |
| CHEXA   | 20          | Pressure  | 242  | -5.3833E-4 | -5.3189E-4       | 1.2       |
| CHEXA   | 20          | Force     | 242  | -2.5352E-3 | -2.4781E-3       | 2.3       |
| CHEXA   | 50          | Pressure  | 242  | -8.0286E-3 | -7.9292E-3       | 1.2       |
| CHEXA   | 50          | Force     | 242  | -3.5738E-2 | -3.4992E-2       | 2.1       |
| CHEXA   | 75          | Pressure  | 242  | -2.6861E-2 | -2.6443E-2       | 1.6       |
| CHEXA   | 75          | Force     | 242  | -1.1837E-1 | -1.1551E-1       | 2.4       |
| CHEXA   | 100         | Pressure  | 36   | -6.3389E-2 | -6.2405E-2       | 1.6       |
| CHEXA   | 100         | Force     | 36   | -2.7794E-1 | -2.7118E-1       | 2.4       |

**\*Note:**  $\lambda$  = length/thickness

**Table 2. Test Case 2 Results (T3 Translation at location C) (m)**

| Filename | $\lambda^*$ | Load Case | Node | SFDM       | Autodesk Nastran | Error (%) |
|----------|-------------|-----------|------|------------|------------------|-----------|
| CQUAD4   | 10          | Pressure  | 1    | -7.8661E-5 | -7.8029E-5       | 0.8       |
| CQUAD4   | 10          | Force     | 1    | -4.1087E-4 | -3.8270E-4       | 6.9       |
| CQUAD4   | 20          | Pressure  | 36   | -5.5574E-4 | -5.4897E-4       | 1.2       |
| CQUAD4   | 20          | Force     | 36   | -2.5946E-3 | -2.4829E-3       | 4.3       |
| CQUAD4   | 50          | Pressure  | 36   | -8.3480E-3 | -8.2171E-3       | 1.6       |
| CQUAD4   | 50          | Force     | 36   | -3.7454E-2 | -3.6097E-2       | 3.6       |
| CQUAD4   | 75          | Pressure  | 36   | -2.8053E-2 | -2.7586E-2       | 1.7       |
| CQUAD4   | 75          | Force     | 36   | -1.2525E-1 | -1.2064E-1       | 3.7       |
| CQUAD4   | 100         | Pressure  | 1    | -6.6390E-2 | -6.5143E-2       | 1.9       |
| CQUAD4   | 100         | Force     | 1    | -2.9579E-1 | -2.8424E-1       | 3.9       |

**\*Note:**  $\lambda$  = length/thickness

## References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLV09/89.

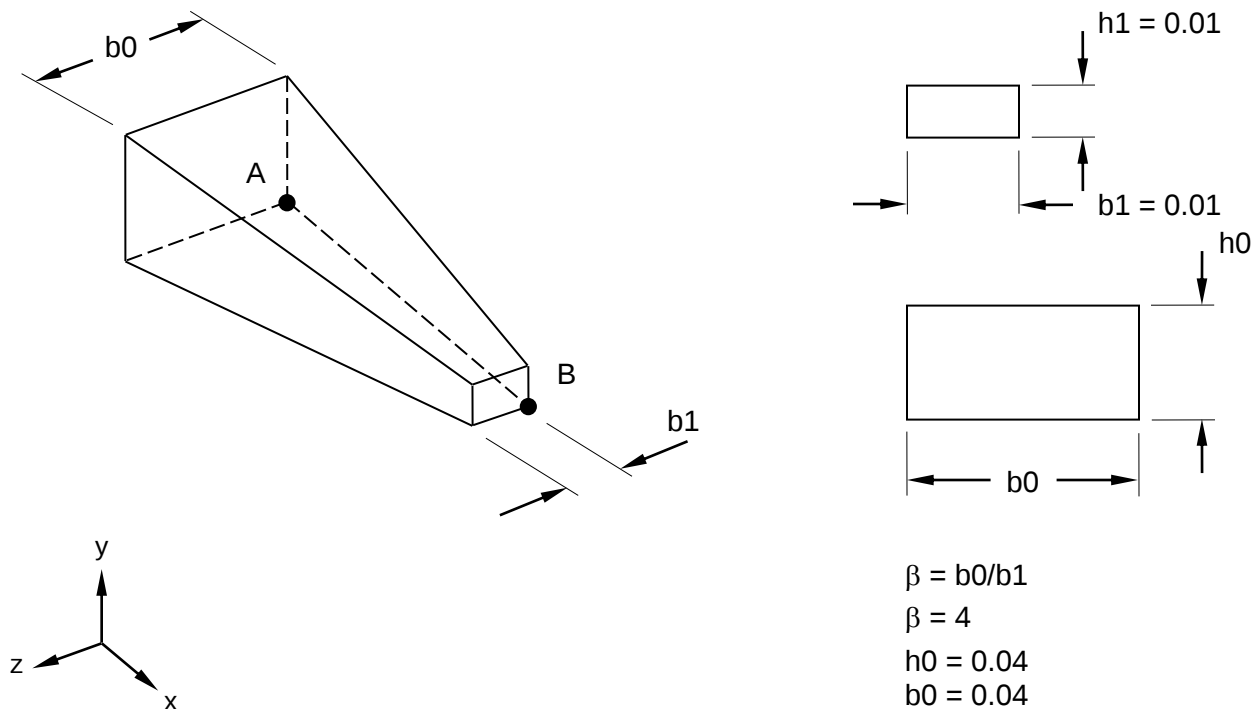
## 6.4 Mechanical Structures – Normal Modes/Eigenvalue Analysis

The following mechanical structures verification problems using standard SFM Benchmarks are performed using normal modes/eigenvalue analysis.

### 6.4.1 Cantilever Beam with a Variable Rectangular Section

#### Problem Description

Figure 1 shows the cantilever beam with a variable rectangular section. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the beam are determined. All dimensions are in meters.



**Figure 1. Cantilever Beam with a Variable Rectangular Section**

#### Autodesk Nastran Analysis Model Filenames

- vm6\_4\_1d.nas – Diagonal mass formulation
- vm6\_4\_1c.nas – Coupled mass formulation

#### Model Data

##### *Finite Element Modeling*

- 11 nodes, 10 beam elements (tapered)

##### *Units*

meter/Newton/second

### Material Properties

Young's Modulus:  $E = 2.0 \text{ E}+11 \text{ Pa}$

Mass Density:  $\rho = 7800 \text{ kg/m}^3$

### Boundary Conditions

Point A (node 1) is fully constrained in all translations and rotations. All other nodes are constrained in the Z-translation and X and Y-rotations only.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Mode Number | $\beta$ | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |         | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 4       | 54.18                  | 54.33                                      | 0.3       | 54.15                                     | 0.1       |
| 2           | 4       | 171.94                 | 170.71                                     | 0.7       | 172.02                                    | 0.2       |
| 3           | 4       | 384.40                 | 373.30                                     | 0.3       | 383.92                                    | 0.1       |
| 4           | 4       | 697.24                 | 657.59                                     | 5.7       | 693.82                                    | 0.5       |
| 5           | 4       | 1,112.28               | 1,017.22                                   | 8.5       | 1,102.04                                  | 0.9       |

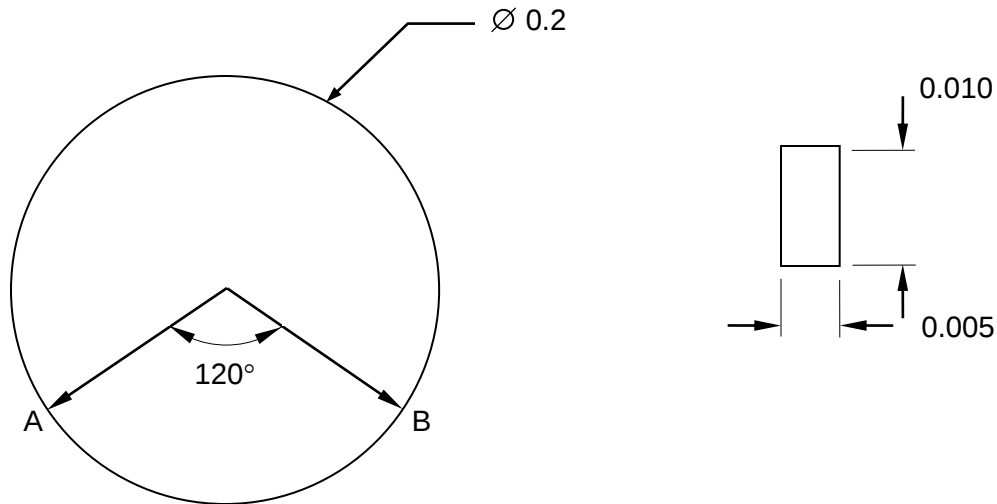
### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLL09/89.

## 6.4.2 Thin Circular Ring Clamped at Two Points

### Problem Description

Figure 1 shows the circular ring. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the circular ring are determined. All dimensions are in meters.



**Figure 1. Thin Circular Ring Clamped at Two Points**

### Autodesk Nastran Analysis Model Filenames

- vm6\_4\_2d.nas – Diagonal mass formulation
- vm6\_4\_2c.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- 29 nodes, 29 bar elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Radius:  $R = 0.1$  m

#### *Cross Sectional Properties*

Rectangular Cross Section = (0.010 m x 0.005 m)

### Material Properties

Young's Modulus:  $E = 7.2 \text{ E}+10 \text{ Pa}$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 2700 \text{ kg/m}^3$

### Boundary Conditions

Points A and B (nodes 1 and 2) are fully constrained in all translations and rotations. All other nodes are constrained the Z-translation and X and Y-rotations only.

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Frequency Results**

|                | Theory                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1              | 235.3                        | 235.9                                         | 0.3       | 236.0                                        | 0.3       |
| 2              | 575.3                        | 576.5                                         | 0.2       | 576.1                                        | 0.1       |
| 3              | 1,105.7                      | 1,106.1                                       | 0.0       | 1,105.6                                      | 0.0       |
| 4              | 1,405.6                      | 1,406.8                                       | 0.0       | 1,405.2                                      | 0.0       |
| 5              | 1,751.1                      | 1,751.4                                       | 0.0       | 1,747.5                                      | 0.2       |
| 6              | 2,557.0                      | 2,556.9                                       | 0.0       | 2,550.0                                      | 0.3       |
| 7              | 2,801.5                      | 2,751.0                                       | 1.8       | 2,774.2                                      | 1.0       |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLL12/89.

### 6.4.3 Vibration Modes of a Thin Pipe Elbow

#### Problem Description

Figure 1 shows the thin pipe elbow. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the thin pipe elbow are determined. All dimensions are in meters.

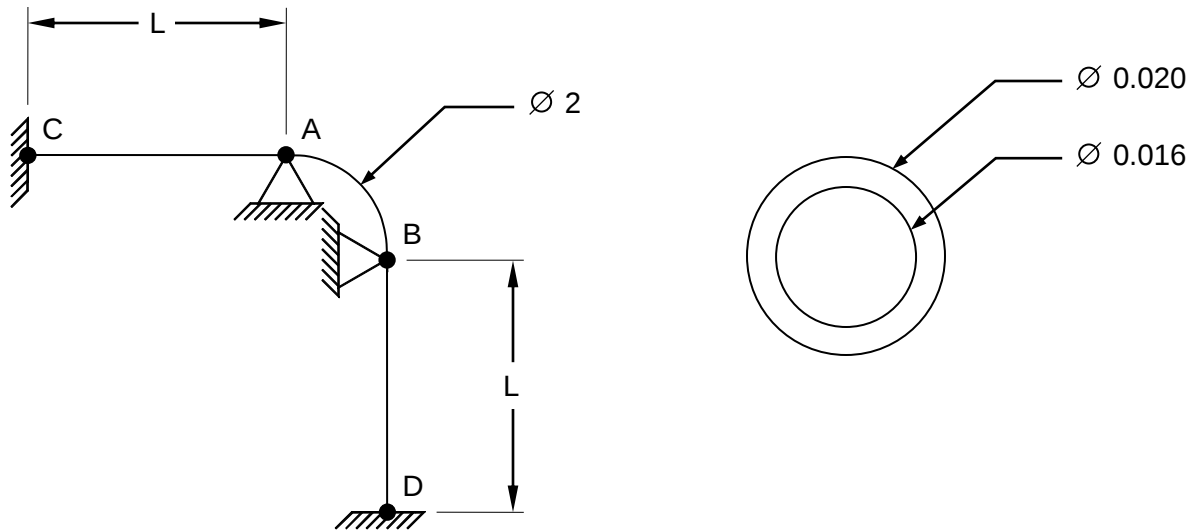


Figure 1. Thin Pipe Elbow

#### Autodesk Nastran Analysis Model Filenames

- vm6\_4\_3ad.nas – Diagonal mass formulation
- vm6\_4\_3ac.nas – Coupled mass formulation
- vm6\_4\_3bd.nas – Diagonal mass formulation
- vm6\_4\_3bc.nas – Coupled mass formulation
- vm6\_4\_3cd.nas – Diagonal mass formulation
- vm6\_4\_3cc.nas – Coupled mass formulation

#### Model Data

##### Finite Element Modeling

- Test 1 (vm6\_4\_3a): L = 0.0 m, 19 nodes, 18 bar elements.
- Test 2 (vm6\_4\_3b): L = 0.6 m, 19 nodes, 18 bar elements.
- Test 3 (vm6\_4\_3c): L = 2.0 m, 29 nodes, 28 bar elements.

##### Units

meter/Newton/second



**Model Geometry**

Radius of Arc:  $R = 1$  m

External Radius of Pipe:  $R_e = 0.010$  m

Inside Radius of Pipe:  $R_i = 0.008$  m

**Material Properties**

Young's Modulus:  $E = 2.1 \text{ E}+11$  Pa

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 7800$  kg/m<sup>3</sup>

**Boundary Conditions**

Test 1 (vm6\_4\_3a): Points C and D (nodes 1 and 2) are fully constrained in all translations and rotations.

Test 2 (vm6\_4\_3b): Points C and D (nodes 1 and 4) are fully constrained in all translations and rotations. Point B (node 2) is constrained in the X and Z-translations. Point C (node 3) is constrained in the Y and Z-translations.

Test 3 (vm6\_4\_3c): Points C and D (nodes 1 and 4) are fully constrained in all translations and rotations. Point B (node 2) is constrained in the X and Z-translations. Point C (node 3) is constrained in the Y and Z-translations.

**Solution Type**

Normal Modes/Eigenvalue – Subspace iterative method

**Comparison of Results**

The tabular results are given in Tables 1, 2 and 3.

**Table 1. Test 1 (vm643a) Frequency Results**

| Mode Number | L | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
|             |   | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 0 | 44                     | 44                                         | 0.0       | 44                                        | 0.0       |
| 2           | 0 | 119                    | 119                                        | 0.0       | 119                                       | 0.0       |
| 3           | 0 | 125                    | 125                                        | 0.0       | 125                                       | 0.0       |
| 4           | 0 | 227                    | 224                                        | 1.3       | 225                                       | 0.9       |

Table 2. Test 2 (vm643b) Frequency Results

|             |   | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | L | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 1 | 33                     | 33                                         | 0.0       | 33                                        | 0.0       |
| 2           | 1 | 94                     | 94                                         | 0.0       | 94                                        | 0.0       |
| 3           | 1 | 100                    | 98                                         | 2.0       | 99                                        | 1.0       |
| 4           | 1 | 180                    | 182                                        | 1.1       | 183                                       | 1.7       |

Table 3. Test 3 (vm643c) Frequency Results

|             |   | NAFEMS                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|---|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | L | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 2 | 17.9                   | 17.6                                       | 1.7       | 17.7                                      | 1.1       |
| 2           | 2 | 24.8                   | 24.4                                       | 1.6       | 24.4                                      | 1.6       |
| 3           | 2 | 25.3                   | 24.9                                       | 1.6       | 24.9                                      | 1.6       |
| 4           | 2 | 27.0                   | 26.6                                       | 1.5       | 26.7                                      | 1.1       |

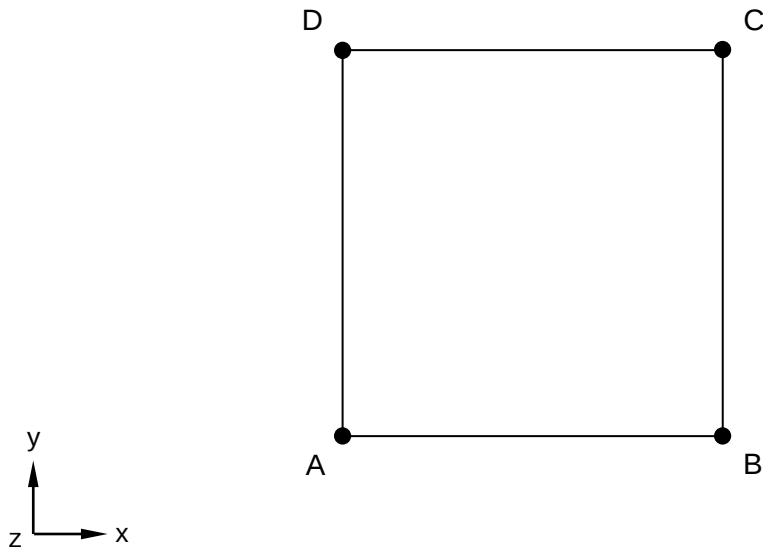
## References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLL14/89.

## 6.4.4 Thin Square Plate (Clamped or Free)

### Problem Description

Figure 1 shows the square plate. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies are determined.



**Figure 1. Thin Square Plate**

### Autodesk Nastran Analysis Model Filenames

- vm6\_4\_4ad.nas – Diagonal mass formulation
- vm6\_4\_4ac.nas – Coupled mass formulation
- vm6\_4\_4bd.nas – Diagonal mass formulation
- vm6\_4\_4bc.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- 121 nodes, 100 5-DOF/node quadrilateral plate elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 1$  m

Width:  $w = 1$  m

### Material Properties

Young's Modulus:  $E = 2.1 \text{ E}+11 \text{ Pa}$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 7800 \text{ kg/m}^3$

### Boundary Conditions

Test 1 (vm6\_4\_4a): The nodes along side AB are constrained in all translations and rotations.

Test 2 (vm6\_4\_4b): Unconstrained plate; only the lower left, lower right, and upper right corner nodes are constrained in all translations and rotations (points A, B and C).

### Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

### Comparison of Results

The tabular results are given in Tables 1 and 2.

**Table 1. Test 1 (vm6\_4\_4a) Frequency Results**

|             | Theory                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 8.73                   | 8.64                                       | 1.0       | 8.68                                      | 0.6       |
| 2           | 21.30                  | 21.04                                      | 1.2       | 21.30                                     | 0.0       |
| 3           | 53.55                  | 52.25                                      | 2.4       | 53.78                                     | 0.4       |
| 4           | 68.30                  | 66.15                                      | 3.1       | 68.86                                     | 0.8       |
| 5           | 77.74                  | 75.82                                      | 2.5       | 78.50                                     | 1.0       |
| 6           | 136.05                 | 131.05                                     | 3.7       | 139.28                                    | 2.4       |

**Table 2. Test 2 (vm6\_4\_4b) Frequency Results**

|             | Theory                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 7           | 33.71                  | 33.10                                      | 1.8       | 33.72                                     | 0.0       |
| 8           | 49.45                  | 47.34                                      | 4.6       | 49.35                                     | 0.2       |
| 9           | 61.05                  | 59.03                                      | 3.3       | 61.38                                     | 0.5       |
| 10, 11      | 87.52                  | 83.84                                      | 4.2       | 88.01                                     | 0.6       |

## References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLS01/89.

### 6.4.5 Simply – Supported Rectangular Plate

#### Problem Description

Figure 1 shows the rectangular plate, with a thickness of 0.01 meters. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the rectangular plate are determined. All dimensions are in meters.

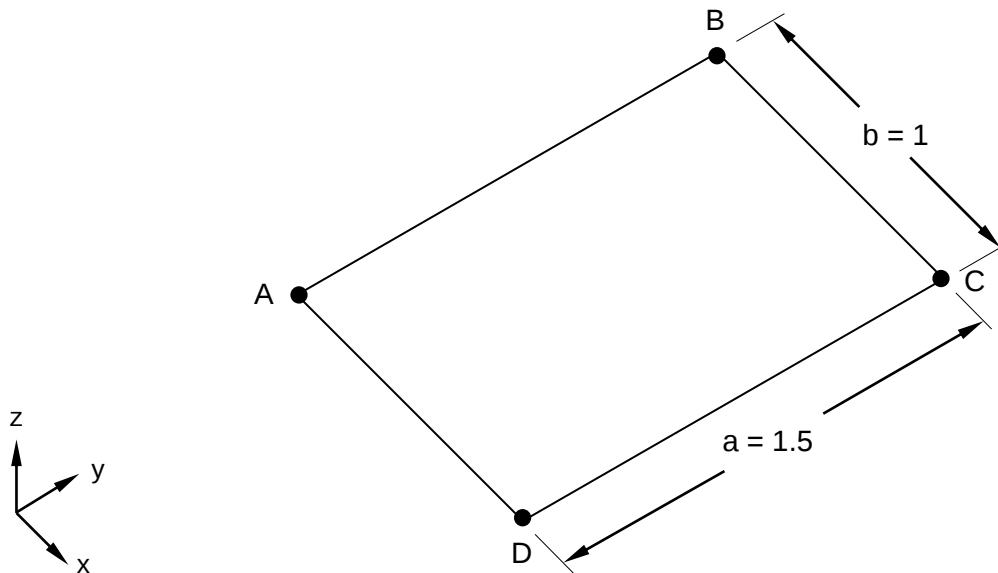


Figure 1. Simply – Supported Rectangular Plate

#### Autodesk Nastran Analysis Model Filenames

- vm6\_4\_5d.nas – Diagonal mass formulation
- vm6\_4\_5c.nas – Coupled mass formulation

#### Model Data

##### *Finite Element Modeling*

- 176 nodes, 150 5-DOF/node quadrilateral plate elements

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L = 1.5$  m

Width:  $w = 1.0$  m

### Material Properties

Young's Modulus:  $E = 2.1 \text{ E}+11 \text{ Pa}$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 7800 \text{ kg/m}^3$

### Boundary Conditions

The nodes on all sides of the plate are constrained in the Z-translation. Nodes 47, 55 and 119 are constrained in all translations and rotations except for the Z-translation.

### Solution Type

Normal Modes/Eigenvalue – Lanczos method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Frequency Results**

|                | Theory                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 4              | 35.63                        | 35.57                                         | 0.2       | 35.99                                        | 1.0       |
| 5              | 68.51                        | 68.39                                         | 0.2       | 69.97                                        | 2.1       |
| 6              | 109.62                       | 109.88                                        | 0.3       | 114.00                                       | 4.0       |
| 7              | 123.32                       | 123.48                                        | 0.1       | 128.64                                       | 4.3       |
| 8              | 142.51                       | 142.38                                        | 0.1       | 149.27                                       | 4.7       |

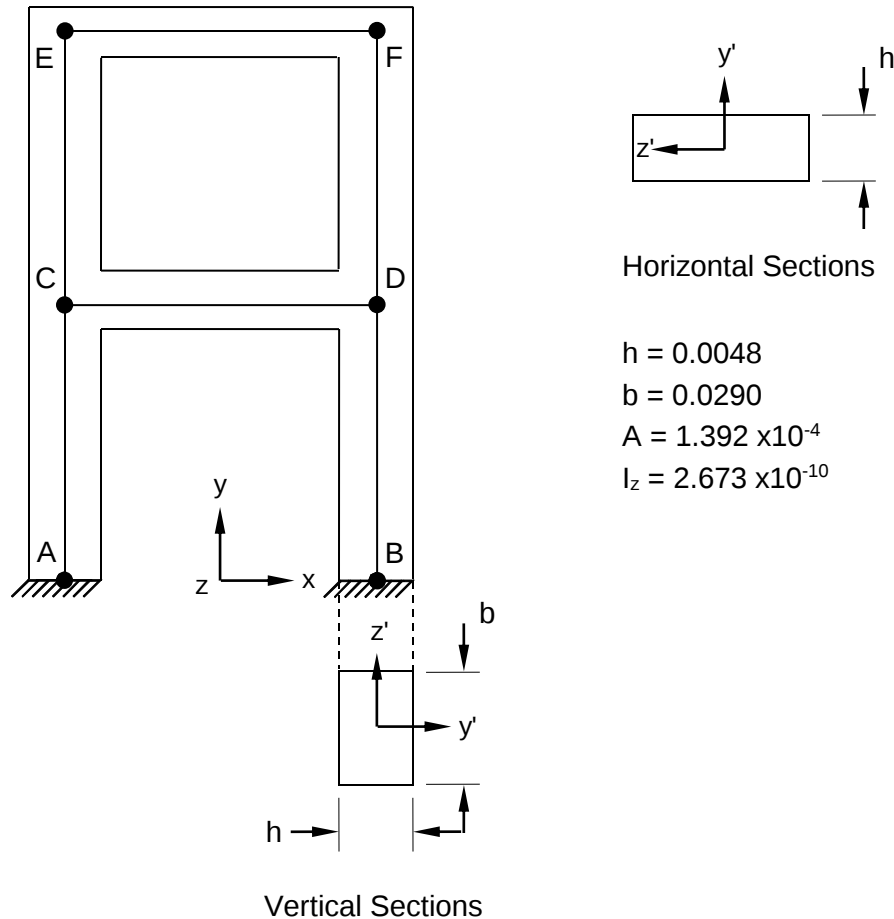
### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLS03/89.

### 6.4.6 Bending of a Symmetric Truss

#### Problem Description

Figure 1 shows the symmetric truss. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the truss are determined. All dimensions are in meters.



**Figure 1. Symmetric Truss**

#### Autodesk Nastran Analysis Model Filenames

- vm6\_4\_6d.nas – Diagonal mass formulation
- vm6\_4\_6c.nas – Coupled mass formulation

#### Model Data

##### *Finite Element Modeling*

- 24 nodes, 24 bar elements



## Units

meter/Newton/second

## Material Properties

Young's Modulus:  $E = 2.1 \text{ E}+11 \text{ Pa}$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 7800 \text{ kg/m}^3$

## Boundary Conditions

Points A and B (nodes 1 and 4) are fully constrained in all translations and rotations. Nodes 2–3 and 5–24 are constrained in the Z-translation and X and Y-rotations.

## Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

## Comparison of Results

The tabular results are given in Table 1.

**Table 1. Frequency Results**

|                | Theory                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 1              | 8.8                          | 8.8                                           | 0.0       | 8.8                                          | 0.0       |
| 2              | 29.4                         | 29.3                                          | 0.3       | 29.4                                         | 0.0       |
| 3              | 43.8                         | 43.8                                          | 0.0       | 43.8                                         | 0.0       |
| 4              | 56.3                         | 56.2                                          | 0.2       | 56.3                                         | 0.0       |
| 5              | 96.2                         | 95.4                                          | 0.2       | 96.2                                         | 0.0       |
| 6              | 102.6                        | 102.4                                         | 0.2       | 102.7                                        | 0.1       |
| 7              | 147.1                        | 146.1                                         | 0.7       | 147.3                                        | 0.1       |
| 8              | 174.8                        | 173.0                                         | 1.0       | 175.3                                        | 0.3       |
| 9              | 178.8                        | 177.3                                         | 0.8       | 179.2                                        | 0.2       |
| 10             | 206.0                        | 202.7                                         | 1.6       | 206.7                                        | 0.3       |
| 11             | 266.4                        | 262.3                                         | 1.6       | 267.9                                        | 0.6       |
| 12             | 320.0                        | 309.6                                         | 3.3       | 322.3                                        | 0.7       |
| 13             | 335.0                        | 321.8                                         | 4.0       | 338.5                                        | 1.0       |

## References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLX01/89.

### 6.4.7 Hovgaard's Problem – Pipes with Flexible Elbows

#### Problem Description

Figure 1 shows the pipe. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the pipe are determined. All dimensions are in meters.

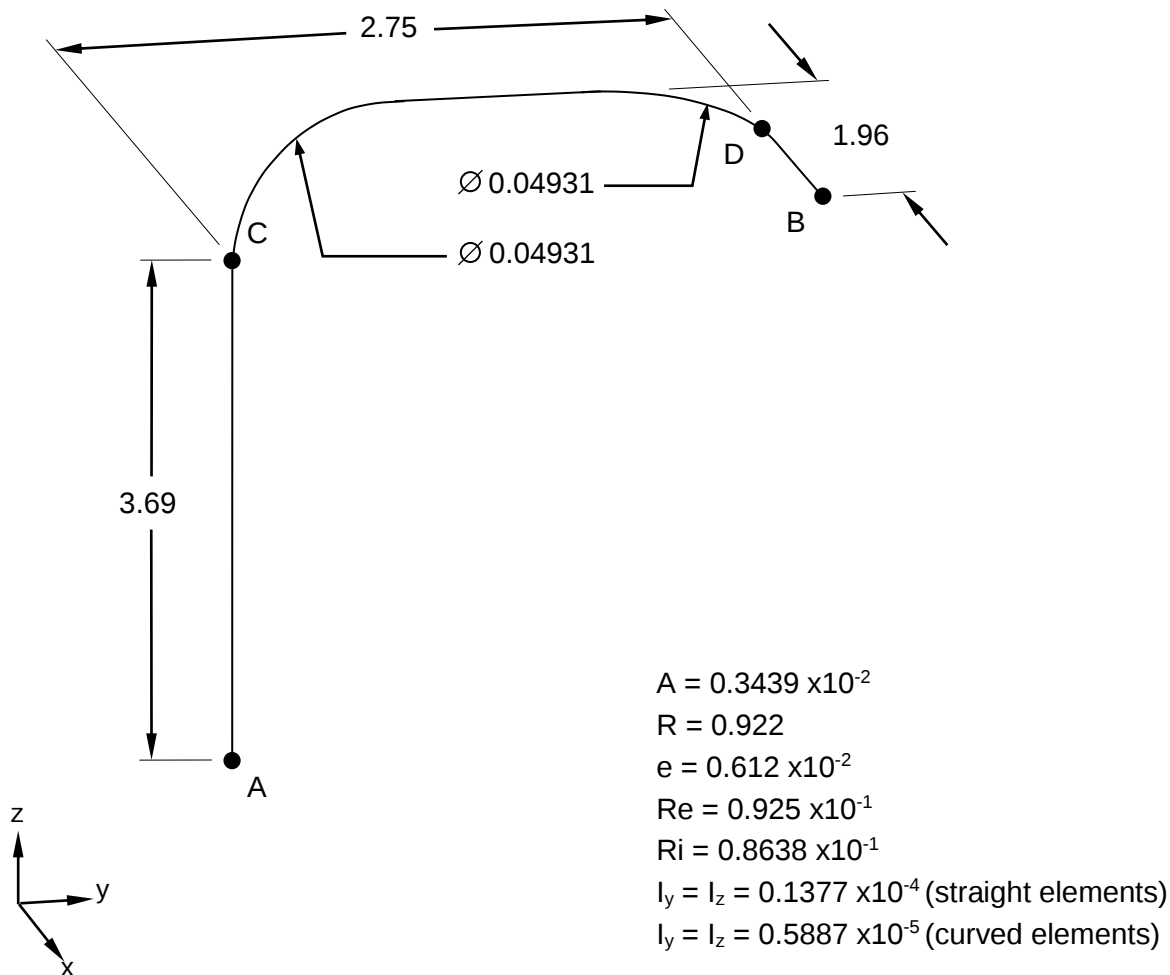


Figure 1. Pipe with Flexible Elbows

#### Autodesk Nastran Analysis Model Filenames

- vm6\_4\_7d.nas – Diagonal mass formulation
- vm6\_4\_7c.nas – Coupled mass formulation

#### Model Data

##### *Finite Element Modeling*

- 26 nodes, 25 bar elements

**Units**

meter/Newton/second

**Material Properties**Young's Modulus:  $E = 1.658 \text{ E}+11 \text{ Pa}$ Poisson's Ratio:  $\nu = 0.3$ Mass Density:  $\rho = 13404.106 \text{ kg/m}^3$ **Boundary Conditions**

Points A and B (nodes 1 and 6) are fully constrained in all translations and rotations.

**Solution Type**

Normal Modes/Eigenvalue – Lanczos method

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Frequency Results**

|             | Theory                 | Autodesk Nastran Diagonal Mass Formulation |           | Autodesk Nastran Coupled Mass Formulation |           |
|-------------|------------------------|--------------------------------------------|-----------|-------------------------------------------|-----------|
| Mode Number | Natural Frequency (Hz) | Natural Frequency (Hz)                     | Error (%) | Natural Frequency (Hz)                    | Error (%) |
| 1           | 10.18                  | 10.52                                      | 3.3       | 10.39                                     | 3.4       |
| 2           | 19.54                  | 20.03                                      | 2.5       | 19.87                                     | 2.5       |
| 3           | 25.47                  | 25.56                                      | 0.4       | 25.35                                     | 0.4       |
| 4           | 48.09                  | 48.46                                      | 0.8       | 47.82                                     | 0.5       |
| 5           | 52.86                  | 52.21                                      | 1.2       | 51.86                                     | 1.3       |
| 6           | 75.94                  | 84.01                                      | 10.6      | 83.16                                     | 10.1      |
| 7           | 80.11                  | 86.27                                      | 7.7       | 85.28                                     | 7.6       |
| 8           | 122.34                 | 127.60                                     | 4.2       | 126.15                                    | 3.5       |
| 9           | 123.15                 | 129.60                                     | 5.2       | 127.97                                    | 4.9       |

**References**

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLX02/89.

## 6.4.8 Rectangular Plates

### Problem Description

Figure 1 shows the rectangular plates. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the rectangular plates are determined. All dimensions are in meters.

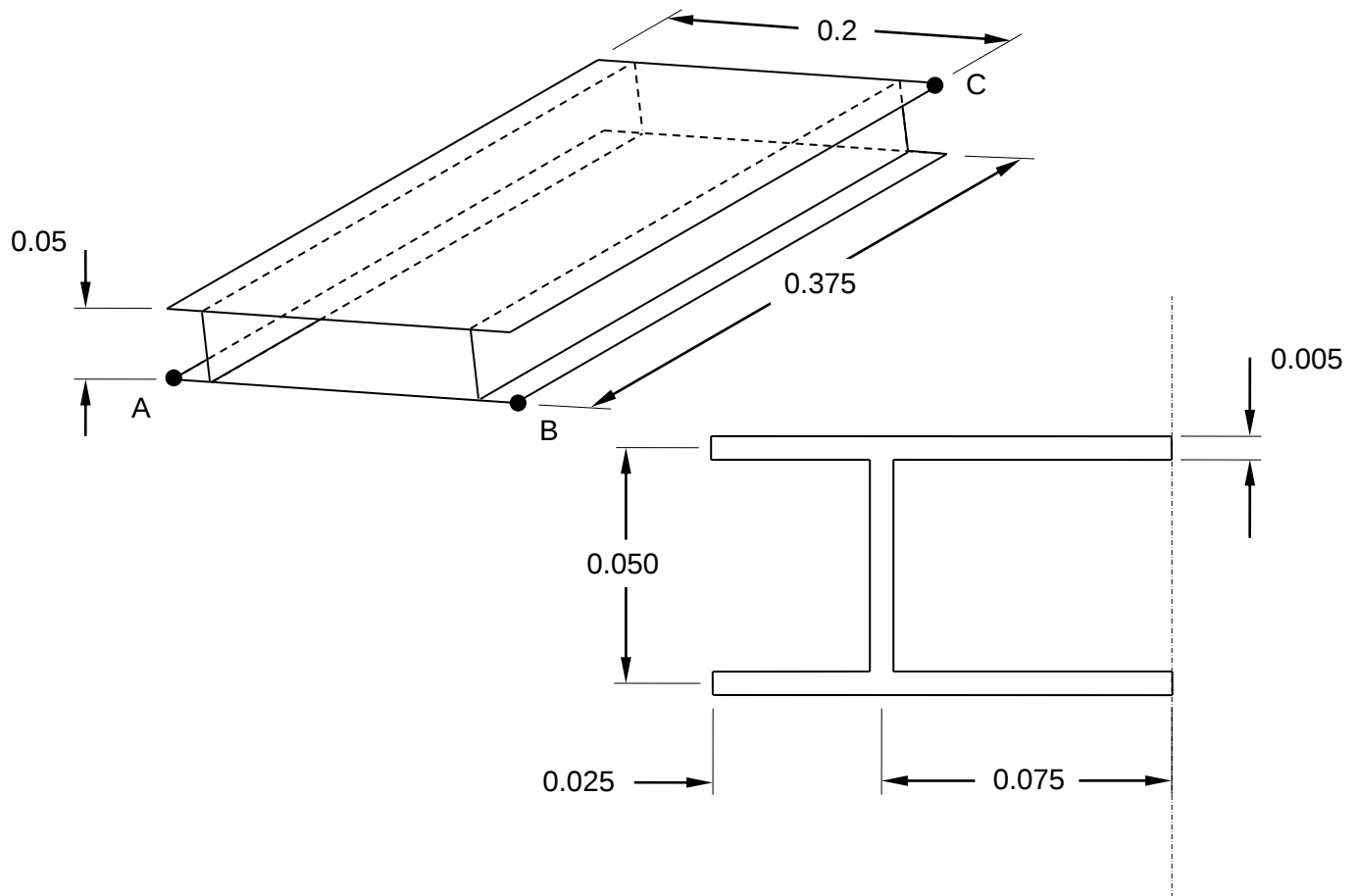


Figure 1. Rectangular Plates

### Autodesk Nastran Analysis Model Filenames

- vm6\_4\_8d.nas – Diagonal mass formulation
- vm6\_4\_8c.nas – Coupled mass formulation

### Model Data

#### *Finite Element Modeling*

- 320 nodes, 300 5-DOF/node quadrilateral plate elements

#### *Units*

meter/Newton/second

### Model Geometry

Length:  $L = 0.375$  m

Width:  $w = 0.200$  m

Thickness:  $t = 0.050$  m

### Material Properties

Young's Modulus:  $E = 2.1 \text{ E}+11$  Pa

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 7800$  kg/m<sup>3</sup>

### Boundary Conditions

Constraint Set 1: Nodes 2, 69, and 84 (points A, B, C) are fully constrained in all translations and rotations.

### Solution Type

Normal Modes/Eigenvalue – Lanczos method

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Frequency Results**

|                | Theory                       | Autodesk Nastran Diagonal<br>Mass Formulation |           | Autodesk Nastran Coupled<br>Mass Formulation |           |
|----------------|------------------------------|-----------------------------------------------|-----------|----------------------------------------------|-----------|
| Mode<br>Number | Natural<br>Frequency<br>(Hz) | Natural<br>Frequency<br>(Hz)                  | Error (%) | Natural<br>Frequency<br>(Hz)                 | Error (%) |
| 7              | 584                          | 584                                           | 0.0       | 594                                          | 1.7       |
| 8              | 826                          | 818                                           | 1.0       | 844                                          | 2.2       |
| 9              | 855                          | 847                                           | 0.9       | 873                                          | 2.1       |
| 10             | 911                          | 901                                           | 1.2       | 932                                          | 2.3       |
| 11             | 1,113                        | 1,087                                         | 2.5       | 1,137                                        | 2.1       |
| 12             | 1,136                        | 1,146                                         | 0.8       | 1,188                                        | 4.6       |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLX03/89.

## 6.5 Stationary Thermal Tests – Steady-State Heat Transfer Analysis

The following stationary thermal verification problems using standard SFM Benchmarks are performed using steady-state heat transfer analysis.

### 6.5.1 L-Plate

#### Problem Description

Figure 1 shows the L-plate. A steady-state heat transfer analysis is performed. The temperature of the L-plate is determined. All dimensions are in meters.

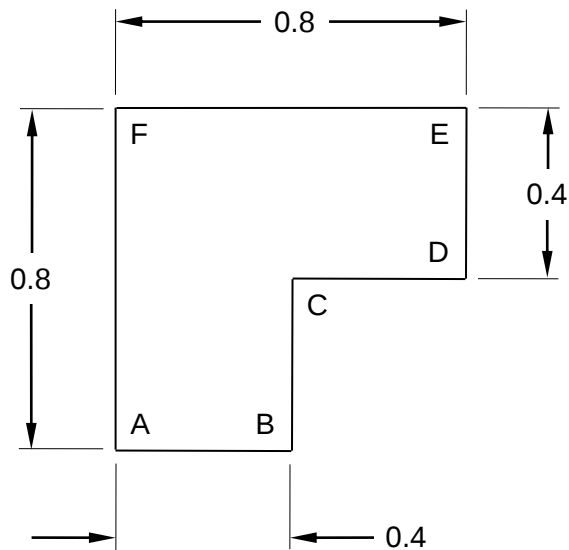


Figure 1. L-Plate

#### Autodesk Nastran Analysis Model Filenames

- vm6\_5\_1.nas

#### Model Data

##### *Finite Element Modeling*

- 21 nodes, 12 5-DOF/node quadrilateral plate elements

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L_1 = 0.8$  m and  $L_2 = 0.4$  m

##### *Material Properties*

$\lambda = 1.0$  W/m °C

### Boundary Conditions

Nodal Temperatures. AF side: temperature is set to 10 °C. DE side: temperature is set to 0 °C.

### Solution Type

Steady-State Heat Transfer

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Temperature Results (°C)**

| Node | Theory | Autodesk Nastran | Error (%) |
|------|--------|------------------|-----------|
| 8    | 7.869  | 7.861            | 0.1       |
| 9    | 5.495  | 5.502            | 0.1       |
| 10   | 2.816  | 2.845            | 1.0       |
| 19   | 8.018  | 8.025            | 0.1       |
| 18   | 5.680  | 5.670            | 0.2       |
| 20   | 2.881  | 2.960            | 2.7       |
| 17   | 8.514  | 8.505            | 0.1       |
| 6    | 6.667  | 6.667            | 0.0       |
| 16   | 2.972  | 2.990            | 0.6       |
| 21   | 9.001  | 9.015            | 0.1       |
| 15   | 8.640  | 8.661            | 0.2       |
| 14   | 9.316  | 9.294            | 0.2       |
| 5    | 9.009  | 8.996            | 0.1       |

### References

1. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. TPLP01/89.

## 7. Buckling Verification Using Theoretical Solutions

The purpose of these buckling test cases is to verify the functionality of Autodesk Nastran using theoretical solutions of well-known engineering buckling problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

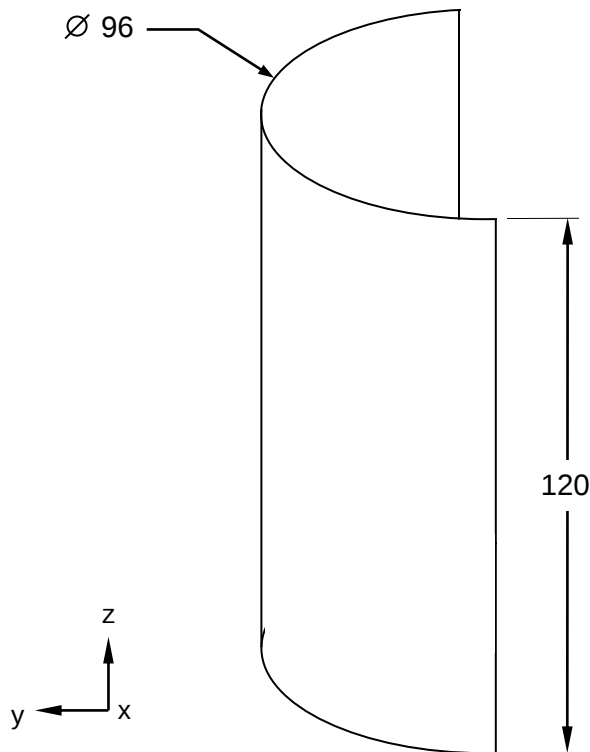
For most cases, discrepancies between Autodesk Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.



## 7.1 Buckling of a Thin Walled Cylinder

### Problem Description

Figure 1 shows the half modeled thin walled cylinder. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in inches.



**Figure 1. Half Cylinder Model**

### Autodesk Nastran Analysis Model Filenames

- vm7\_1a.nas – Test 1
- vm7\_1b.nas – Test 2
- vm7\_1c.nas – Test 3

### Model Data

#### *Finite Element Modeling*

- Test 1 (vm7\_1a): 861 nodes, 800 5-DOF/node quadrilateral plate elements
- Test 2 (vm7\_1b): 3321 nodes, 3200 5-DOF/node quadrilateral plate elements
- Test 3 (vm7\_1c): 13041 nodes, 12800 5-DOF/node quadrilateral plate elements

**Units**

inch/pound/second

**Model Geometry**Length:  $L = 120$  inRadius:  $R = 48$  in**Material Properties**Young's Modulus:  $E = 10.0 \text{ E}+6$  psiPoisson's Ratio:  $\nu = 0.3$ **Boundary Conditions**

Cylindrical coordinates are used for the boundary conditions in which Z-axis is running vertically through the center of the cylinder, R-axis is projected outward normal to the nodes, and T-axis is tangential to the elements. The half model is constrained along the sliced edges using symmetry boundary conditions. One end of the cylinder is constrained in all translations. The other end is constrained in the R and T-translations only. A 1 psi pressure load is applied to all elements acting normal to the elements and toward the center of the cylinder. This acts as a crushing load. A separate load set is created and a downward nodal load is applied along all the nodes on the top edge of the cylinder of 1000 lb and 500 lb for the two end nodes. These nodal forces are applied to the end of the cylinder which contains the R and T-translation constraints.

**Solution Type**

Buckling

**Comparison of Results**

Results are calculated for full cylinder model.

The tabular results are given in Table 1.

**Table 1. Results**

| Description                           | Theory  | Autodesk Nastran | Error (%) |
|---------------------------------------|---------|------------------|-----------|
| Critical Buckling Load (Test 1) (lbs) | 380,276 | 382,746          | 0.6       |
| Critical Buckling Load (Test 2) (lbs) | 380,276 | 382,496          | 0.6       |
| Critical Buckling Load (Test 3) (lbs) | 380,276 | 379,680          | 0.2       |

**References**

1. NASA SP-8007 Buckling of Thin-Walled Circular Cylinders.

## 7.2 Buckling of a Bar with Hinged Ends

### Problem Description

Figure 1 shows the bar model. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in inches.

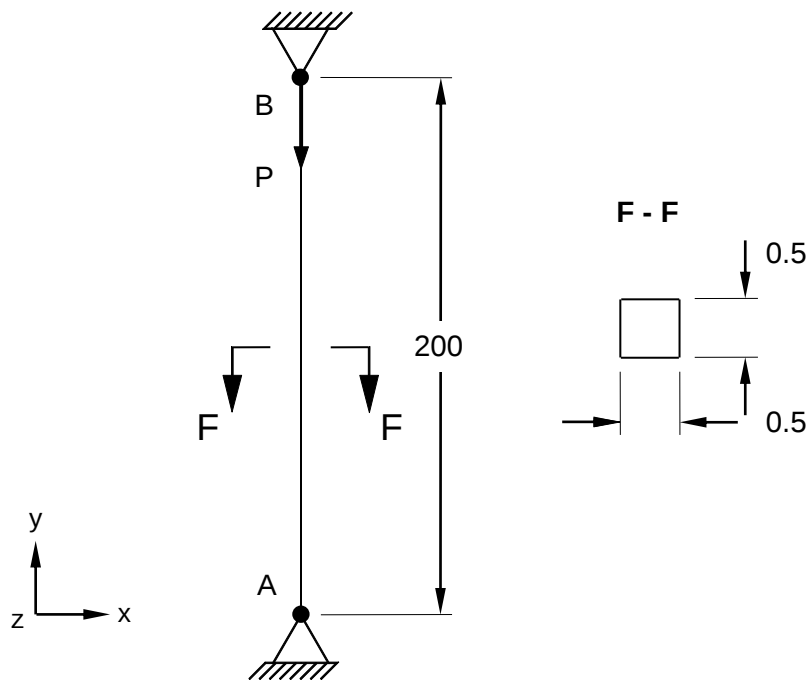


Figure 1. Bar Model

### Autodesk Nastran Analysis Model Filenames

- vm7\_2.nas

### Model Data

#### *Finite Element Modeling*

- 11 nodes, 10 6-DOF/node beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 200$  in

Width:  $w = 0.5$  in

Thickness:  $t = 0.5$  in

**Cross Sectional Properties**

Area:  $A = 0.25 \text{ in}^2$

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+6 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

Supports are pinned: the bottom node is constrained in all translations, and in the Y-rotation, while the upper node is constrained in the X and Z-translations. A unit load P is applied at upper end in the negative Y-direction.

**Solution Type**

Buckling

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                  | Theory | Autodesk Nastran | Error (%) |
|------------------------------|--------|------------------|-----------|
| Critical Buckling Load (lbs) | 38.553 | 38.551           | 0.0       |

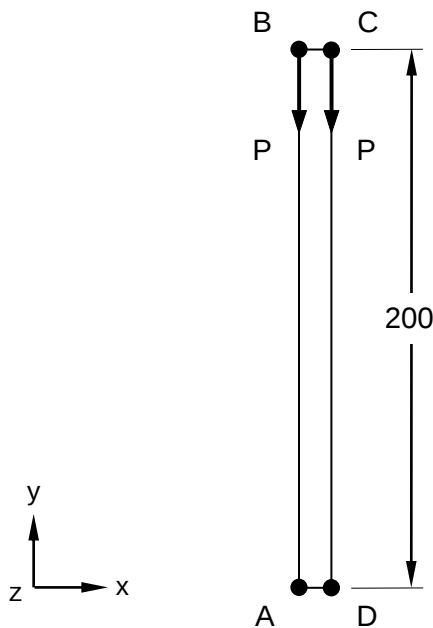
**References**

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1956.

### 7.3 Buckling of a Bar with Hinged Ends Using Plates

#### Problem Description

Figure 1 shows the bar model using plates. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in inches.



**Figure 1. Bar with Hinged Ends Using Plates**

#### Autodesk Nastran Analysis Model Filenames

- vm7\_3.nas

#### Model Data

##### *Finite Element Modeling*

- 42 nodes, 20 5-DOF/node quadrilateral plate elements

##### *Units*

inch/pound/second

##### *Model Geometry*

Length:  $L = 200$  in

Width:  $w = 0.5$  in

Thickness:  $t = 0.5$  in

**Material Properties**

*Young's Modulus:*  $E = 30.0 \text{ E}+6 \text{ psi}$

*Poisson's Ratio:*  $\nu = 0.3$

**Boundary Conditions**

Supports are pinned: the bottom left node (point A) is constrained in all translations, and in the Y-rotation, while the bottom right node (point D) is constrained in the Y and Z-translation and the Y-rotation. The top left node (point B) is constrained in the X and Z-translations, while the top right node (point C) is constrained in the Z-translation. A half unit load  $P$  is applied at each node at the upper end in the negative Y-direction.

**Solution Type**

Buckling

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                  | Theory | Autodesk Nastran | Error (%) |
|------------------------------|--------|------------------|-----------|
| Critical Buckling Load (lbs) | 38.553 | 38.639           | 0.2       |

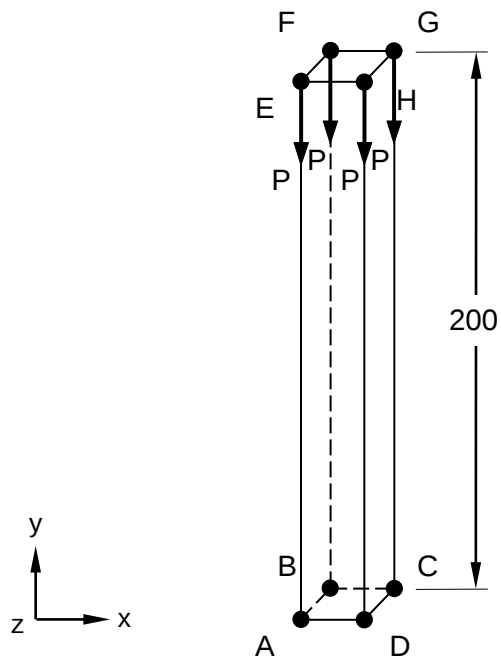
**References**

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1956.

## 7.4 Buckling of a Bar with Hinged Ends Using Solids

### Problem Description

Figure 1 shows the bar model using solids. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in inches.



**Figure 1. Bar with Hinged Ends Using Solids**

### Autodesk Nastran Analysis Model Filenames

- vm7\_4.nas

### Model Data

#### *Finite Element Modeling*

- 84 nodes, 20 3-DOF/node brick elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 200$  in

Width:  $w = 0.5$  in

Thickness:  $t = 0.5$  in

**Cross Sectional Properties**

Area:  $A = 0.25 \text{ in}^2$

**Material Properties**

Young's Modulus:  $E = 30.0 \text{ E}+6 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

Supports are pinned: the bottom left hand side nodes (points A and B) are constrained in all translations, while the upper left hand side nodes (points E and F) are constrained in the X and Z-translations. A quarter unit load P is applied at each node at the upper end in the negative Y-direction.

**Solution Type**

Buckling

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                  | Theory | Autodesk Nastran | Error (%) |
|------------------------------|--------|------------------|-----------|
| Critical Buckling Load (lbs) | 38.553 | 38.724           | 0.4       |

**References**

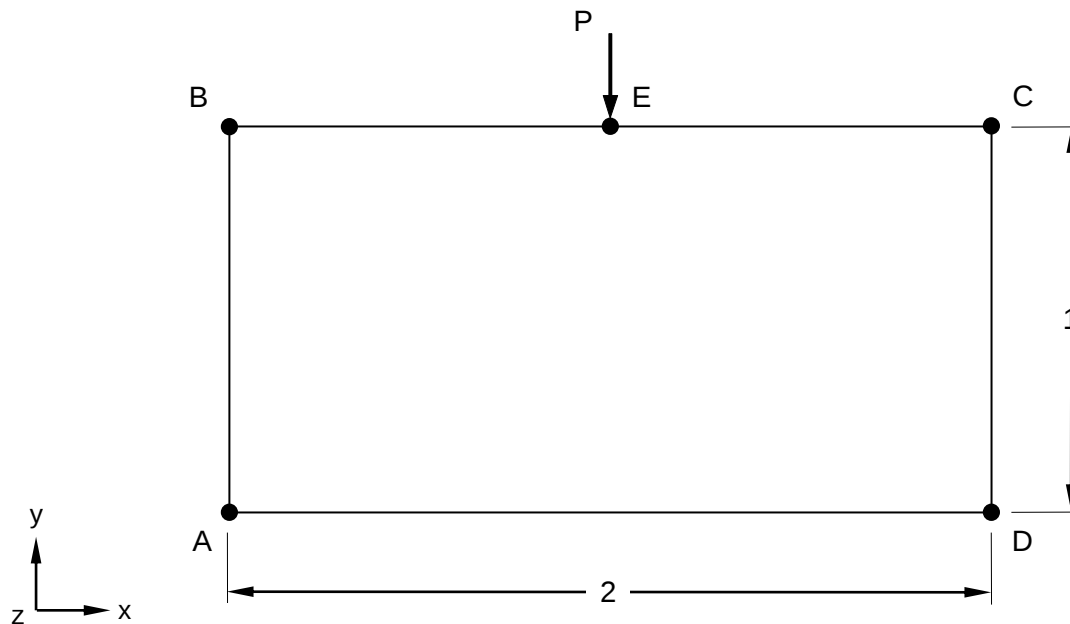
1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1956.



## 7.5 Buckling of a Rectangular Plate Under Concentrated Center Loads

### Problem Description

Figure 1 shows the plate model. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in meters.



**Figure 1. Plate Model**

### Autodesk Nastran Analysis Model Filenames

- vm7\_5.nas

### Model Data

#### *Finite Element Modeling*

- 84 nodes, 72 5-DOF/node quadrilateral plate elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 2.0$  m

Width:  $w = 1.0$  m

Thickness:  $t = 0.01$  m

**Material Properties**

Young's Modulus:  $E = 200.0$  GPa

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

Point A is constrained in all translations and rotations. Points B and C are constrained in the Z-translation and all rotations. Point D is constrained in Y and Z-translations and all rotations. All other nodes on edges AB and CD are constrained in Z-translation, X and Z-rotations. All other nodes on edge BC are constrained in Z-translation, Y and Z-rotations. All other nodes on edge AD are constrained in Y and Z-translations, X and Z-rotations. A uniform load  $P$  of 1.0 kN/m at center of edge BC (point E) is applied in the negative Y-direction.

**Solution Type**

Buckling

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                 | Theory | Autodesk Nastran | Error (%) |
|-----------------------------|--------|------------------|-----------|
| Critical Buckling Load (kN) | 330    | 325              | 1.5       |

**References**

1. Timoshenko, S. and Gere, J. M., *Theory of Elastic Stability*, 2<sup>nd</sup> Edition. New York: McGraw-Hill Book Co., Inc., 1961.

## 7.6 Buckling of a Rectangular Plate Under End Uniform Load

### Problem Description

Figure 1 shows the plate model. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in meters.

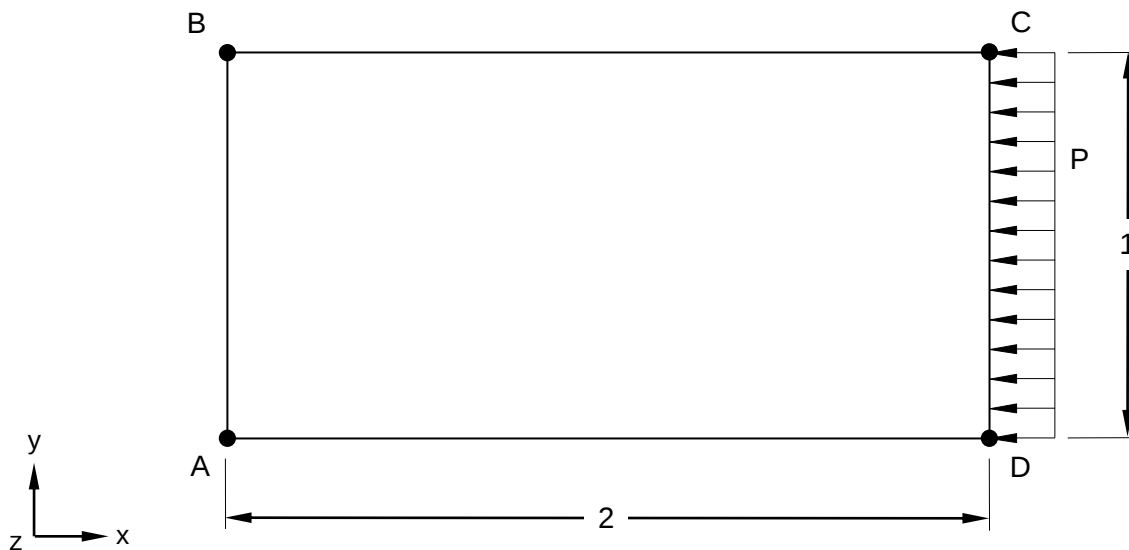


Figure 1. Plate Model

### Autodesk Nastran Analysis Model Filenames

- vm7\_6.nas

### Model Data

#### *Finite Element Modeling*

- 91 nodes, 72 5-DOF/node quadrilateral plate elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 2.0$  m

Width:  $w = 1.0$  m

Thickness:  $t = 0.01$  m

**Material Properties**

Young's Modulus:  $E = 200.0$  GPa

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

Point A is constrained in all translations and rotations. Point B is constrained in the X and Z-translations, and all rotations. Point C is constrained in the Z-translation and all rotations. Point D is constrained in Y and Z-translations, and all rotations. All other nodes on edge AB are constrained in X and Z-translations and X and Z-rotations, while all other nodes on edge CD are constrained in Z-translation, and X and Z-rotations. All other nodes on edge BC are constrained in Z-translation, Y and Z-rotations. All other nodes on edge AD are constrained in Y and Z-translations, Y and Z-rotations. A uniform load  $P = 6.0$  kN at edge CD is applied in the negative X-direction.

**Solution Type**

Buckling

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                 | Theory | Autodesk Nastran | Error (%) |
|-----------------------------|--------|------------------|-----------|
| Critical Buckling Load (kN) | 723    | 723              | 0.0       |

**References**

1. Young, W. C., *Roark's Formulas for Stress and Strain*, 6<sup>th</sup> Edition. New York: McGraw-Hill Co., 1989.

## 8. Dynamics Verification Using Standard NAFEMS Benchmarks

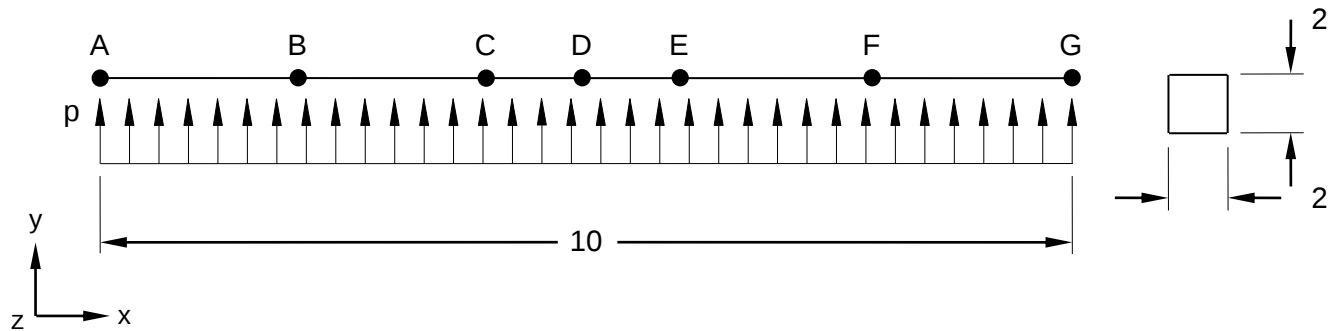
The purpose of these dynamic test cases is to verify the functionality of Autodesk Nastran using standard benchmarks published by NAFEMS (National Agency for Finite Element Methods and Standards, National Engineering Laboratory, Glasgow, U.K.).

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

## 8.1 Transient Forced Vibration Response – Deep Simply Supported Beam

### Problem Description

Figure 1 shows the beam model. A transient dynamic time history analysis is performed on the model. Peak deflection and stress are determined. All dimensions are in meters.



**Figure 1. Beam Cross-Section Model**

### Autodesk Nastran Analysis Model Filenames

- vm8\_1.nas

### Model Data

#### *Finite Element Modeling*

- 7 nodes, 6 6-DOF/node beam elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

#### *Cross Sectional Properties*

Area:  $A = 4$  m<sup>2</sup>

Square Cross Section = (2.0 m x 2.0 m)

#### *Material Properties*

Young's Modulus:  $E = 200.0$  E+6 MPa

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 8000$  kg/m<sup>3</sup>

**Boundary Conditions**

The beam is loaded with a suddenly applied distributed step load  $p = 1 \text{ E}+6 \text{ N/m}$ . One end of the beam is fully constrained in all translations and rotations except the Z-rotation. The other end is constrained in the Y and Z-translations, and X and Y- rotations. A 2% critical damping is also applied.

**Dynamic Parameters**

Number of Time Steps: 1000

Time per Step: 0.0001

Output Intervals: 2

**Solution Type**

Transient Dynamic/Time History

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                      | Theory  | Autodesk Nastran | Error (%) |
|----------------------------------|---------|------------------|-----------|
| Peak Displacement (mm)           | 1.043   | 1.030            | 1.2       |
| Time of Peak Displacement (s)    | 1.17E+1 | 1.18E+1          | 0.9       |
| Peak Stress (N/mm <sup>2</sup> ) | 18.76   | 18.78            | 0.1       |

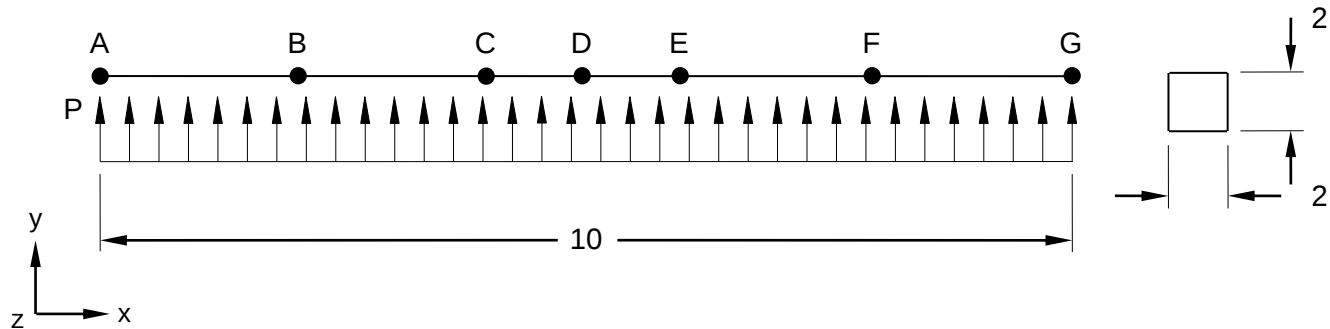
**References**

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Forced Vibration*. Glasgow: NAFEMS, Mar., 1993. Test No. 5T.

## 8.2 Periodic Forced Vibration Response – Deep Simply Supported Beam

### Problem Description

Figure 1 shows the beam model. A transient dynamic time history analysis is performed on the model. Peak deflection and translation are determined. All dimensions are in meters.



**Figure 1. Beam Cross-Section Model**

### Autodesk Nastran Analysis Model Filenames

- vm8\_2.nas

### Model Data

#### *Finite Element Modeling*

- 7 nodes, 6 6-DOF/node beam elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

#### *Cross Sectional Properties*

Area:  $A = 4$  m<sup>2</sup>

Square Cross Section = (2.0 m x 2.0 m)

#### *Material Properties*

Young's Modulus:  $E = 200.0$  E+6 MPa

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 8000$  kg/m<sup>3</sup>



**Boundary Conditions**

The beam is loaded with a steady state periodic function acting as a distributed load of the form

$$P = P_o(\sin \omega t - \sin 3\omega t)$$

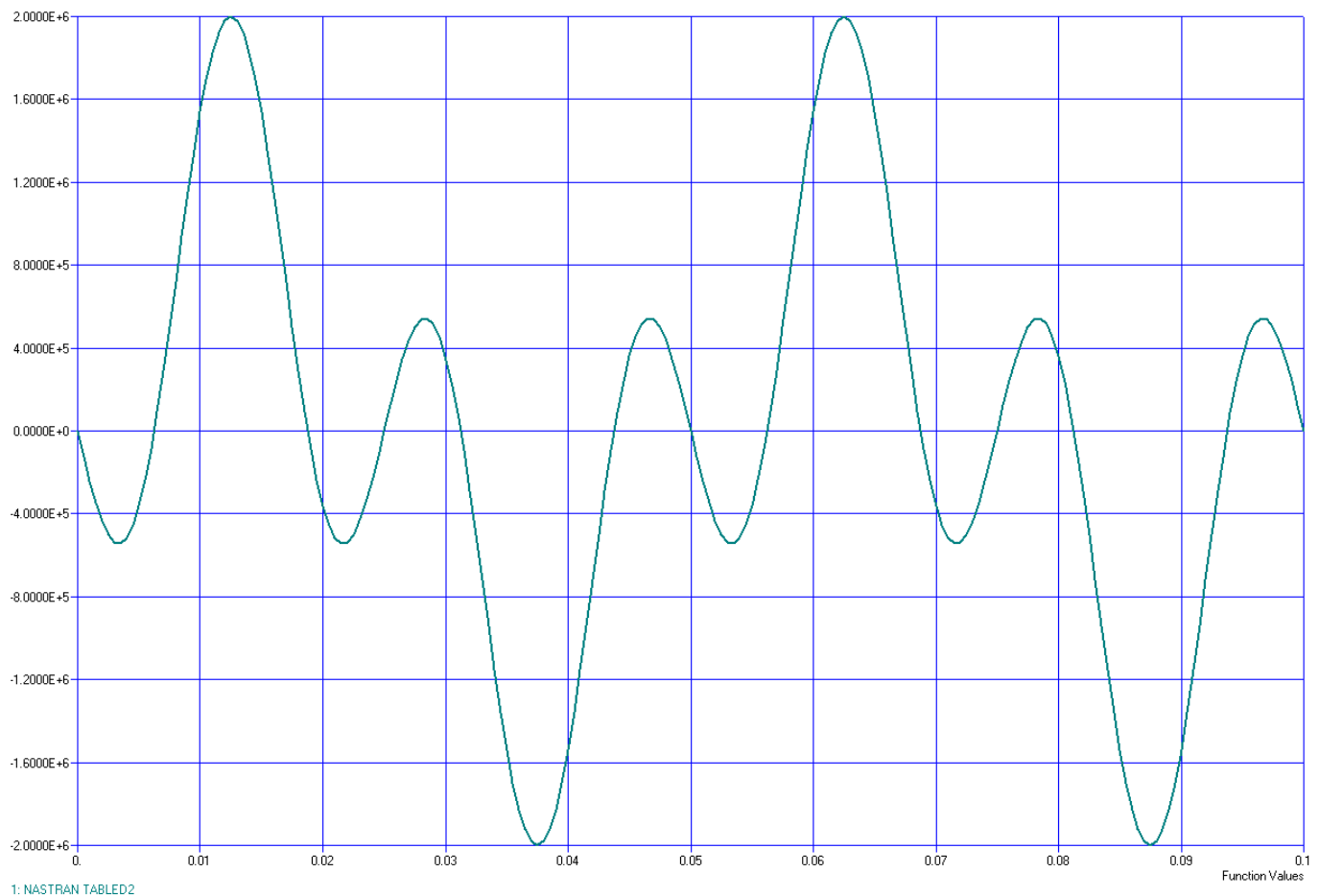
where:

$$P_o = 10^6 \text{ N / m}$$

$$\omega = 2\pi f$$

$$f = 20\text{Hz}$$

The input function is shown in Figure 2.



**Figure 2. Input Loading Function**

One end of the beam is fully constrained in all translations and rotations except the Z-rotation. The other end is constrained in the Y and Z- translations, and X and Y-rotations. A 2% critical damping is also applied.

**Dynamic Parameters**

Number of Time Steps: 5000

Time per Step: 0.0005

Output Intervals: 5

**Solution Type**

Transient Dynamic/Time History

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                      | Theory | Autodesk Nastran | Error (%) |
|----------------------------------|--------|------------------|-----------|
| Peak Displacement (mm)           | 0.951  | 0.934            | 1.8       |
| Peak Stress (N/mm <sup>2</sup> ) | 16.97  | 17.16            | 1.1       |

**References**

1. NAFEMS, *Selected Benchmarks for Forced Vibration*, (J. Maguire, D.J. Dawswell, L. Gould) Test No. 5P.

### 8.3 Modal Transient Forced Vibration Response – Simply Supported Plate

#### Problem Description

Figure 1 shows the plate model. A transient dynamic time history analysis is performed on the model. Peak deflection and stress are determined. All dimensions are in meters.

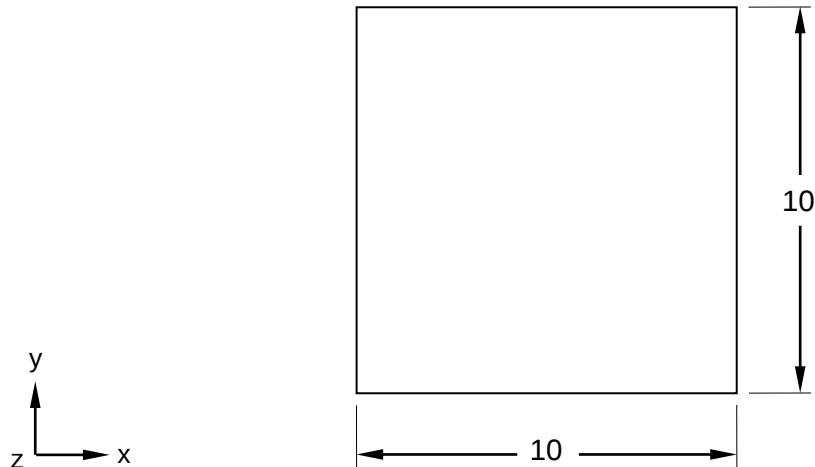


Figure 1. Plate Model

#### Autodesk Nastran Analysis Model Filename

- vm8\_3.nas

#### Model Data

##### *Finite Element Modeling*

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

##### *Units*

meter/Newton/second

##### *Model Geometry*

Length:  $L = 10.0$  m

Width:  $w = 10.0$  m

Thickness:  $t = 0.05$  m

##### *Cross Sectional Properties*

Area:  $A = 100.0$  m<sup>2</sup>

Square Cross Section = (10.0 m x 10.0 m)

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

### Boundary Conditions

The boundary conditions are as follows: fixed  $T_x$ ,  $T_y$ , and  $R_z$  at all nodes; fixed  $T_z$  along all 4 edges; fixed  $R_x$  along edges  $X = 0$  and  $X = 10 \text{ m}$  (left and right edges); fixed  $R_y$  along  $Y = 0$  and  $Y = 10 \text{ m}$  (top and bottom edges). The plate is loaded with a suddenly applied step load acting as a pressure load of  $100 \text{ N/m}^2$  over the whole plate.

### Dynamic Parameters

Number of time steps: 200

Time per step: 0.002

Output Intervals: 0

### Solution Type

Transient Dynamic/Time History

### Solution Method

Modal Transient

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                      | Theory | Autodesk Nastran | Error (%) |
|----------------------------------|--------|------------------|-----------|
| Peak Displacement (mm)           | 3.523  | 3.436            | 2.5       |
| Time of Peak Displacement (s)    | 0.210  | 0.210            | 0.0       |
| Peak Stress (N/mm <sup>2</sup> ) | 2.484  | 2.476            | 0.3       |

### References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

## 8.4 Harmonic Forced Vibration Response – Simply Supported Plate

### Problem Description

Figure 1 shows the plate model. A harmonic forced vibration response analysis is performed on the model. Peak deflection and stress are determined, as well as frequencies. All dimensions are in meters.

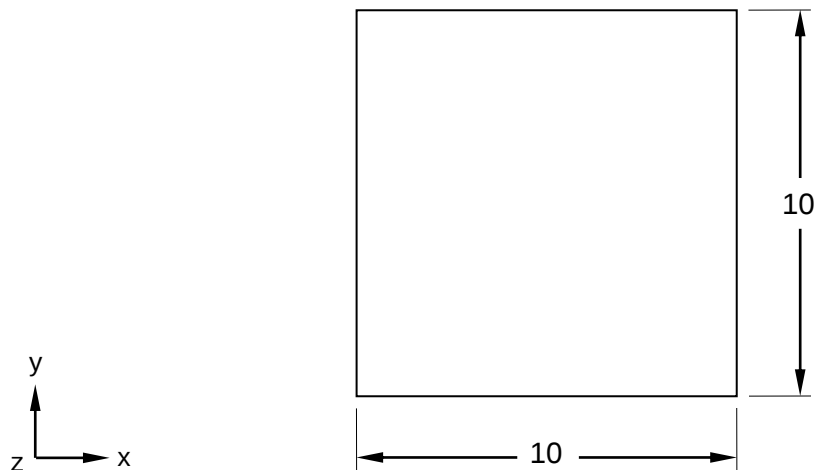


Figure 1. Plate Model

### Autodesk Nastran Analysis Model Filename

- vm8\_4.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

Width:  $w = 10.0$  m

Thickness:  $t = 0.05$  m

#### *Cross Sectional Properties*

Area:  $A = 100.0$  m<sup>2</sup>

Square Cross Section = (10.0 m x 10.0 m)

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

### Boundary Conditions

The boundary conditions are as follows: fixed  $T_x$ ,  $T_y$ , and  $R_z$  at all nodes; fixed  $T_z$  along all 4 edges; fixed  $R_x$  along edges  $X = 0$  and  $X = 10 \text{ m}$  (left and right edges); fixed  $R_y$  along  $Y = 0$  and  $Y = 10 \text{ m}$  (top and bottom edges). A step loading of magnitude  $100 \text{ N/m}^2$  is applied to the entire plate, as pressure load. Damping is 2% critical across all modes. A frequency dependent function will be applied where frequency range from 0 to 4.16 Hz. The effects of gravity are not considered in this model.

### Dynamic Parameters

Number of Modes: 16

### Solution Type

Frequency/Harmonic Response

### Solution Method

Modal Frequency

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                      | Theory | Autodesk Nastran | Error (%) |
|----------------------------------|--------|------------------|-----------|
| Peak Displacement (mm)           | 45.42  | 45.25            | 0.4       |
| Frequency (Hz)                   | 2.377  | 2.377            | 0.0       |
| Peak Stress (N/mm <sup>2</sup> ) | 30.03  | 31.64            | 5.1       |

### References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

## 8.5 Random Forced Vibration Response – Simply Supported Plate

### Problem Description

Figure 1 shows the plate model. A random forced vibration response analysis is performed on the model. Peak deflection and PSD stress are determined, as well as frequencies. All dimensions are in meters.

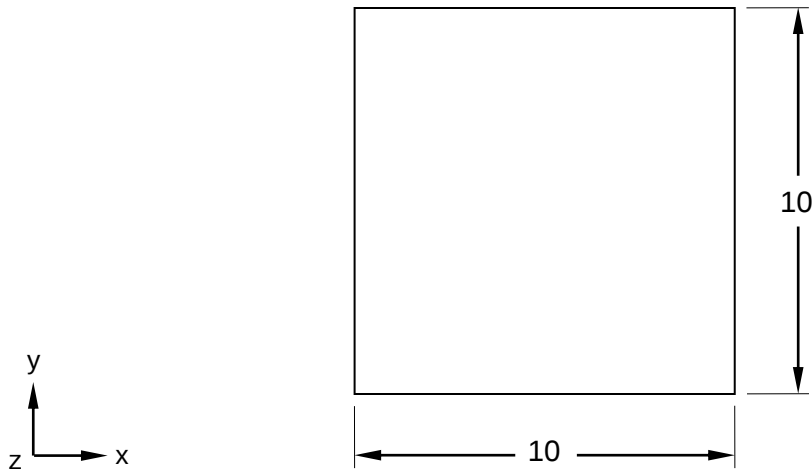


Figure 1. Plate Model

### Autodesk Nastran Analysis Model Filename

- vm8\_5.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

Width:  $w = 10.0$  m

Thickness:  $t = 0.05$  m

#### *Cross Sectional Properties*

Area:  $A = 100.0$  m<sup>2</sup>

Square Cross Section = (10.0 m x 10.0 m)

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

### Boundary Conditions

The boundary conditions are as follows: fixed  $T_x$ ,  $T_y$ , and  $R_z$  at all nodes; fixed  $T_z$  along all 4 edges; fixed  $R_x$  along edges  $X = 0$  and  $X = 10 \text{ m}$  (left and right edges); fixed  $R_y$  along  $Y = 0$  and  $Y = 10 \text{ m}$  (top and bottom edges). A random forcing with uniform power spectral density (of force) PSD is applied to the entire plate ( $\text{PSD} = (100 \text{ N/m}^2)^2/\text{Hz}$ ). Damping is 2% critical across all modes. A frequency dependent function will be applied where frequency range from 0 to 4.16 Hz. The effects of gravity are not considered in this model.

### Dynamic Parameters

Number of Modes: 16

### Solution Type

Random Response

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                          | Theory   | Autodesk Nastran | Error (%) |
|------------------------------------------------------|----------|------------------|-----------|
| Peak Displacement PSD ( $\text{mm}^2/\text{Hz}$ )    | 2,063.20 | 2,047.86         | 0.7       |
| Frequency (Hz)                                       | 2.377    | 2.377            | 0.0       |
| Peak Stress PSD ( $\text{N/mm}^2$ ) <sup>2</sup> /Hz | 1,025.44 | 1,000.80         | 2.4       |

### References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.



## 8.6 Direct Transient Forced Vibration Response – Simply Supported Plate

### Problem Description

Figure 1 shows the plate model. Two transient dynamic time history analyses are performed on the model, one with structural damping and another one with Rayleigh damping. Peak deflection and stress are determined. All dimensions are in meters.

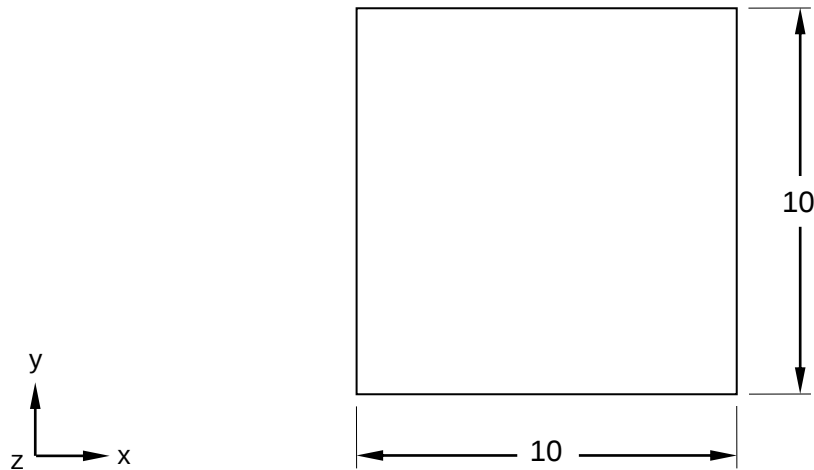


Figure 1. Plate Model

### Autodesk Nastran Analysis Model Filename

- vm8\_6.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

Width:  $w = 10.0$  m

Thickness:  $t = 0.05$  m

#### *Cross Sectional Properties*

Area:  $A = 100.0$  m<sup>2</sup>

Square Cross Section = (10.0 m x 10.0 m)

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

### Boundary Conditions

The boundary conditions are as follows: fixed  $T_x$ ,  $T_y$ , and  $R_z$  at all nodes; fixed  $T_z$  along all 4 edges; fixed  $R_x$  along edges  $X = 0$  and  $X = 10 \text{ m}$  (left and right edges); fixed  $R_y$  along  $Y = 0$  and  $Y = 10 \text{ m}$  (top and bottom edges). A time varying pressure load of  $100 \text{ N/m}^2$  is applied on the entire plate. An overall damping coefficient ( $G$ ) of 0.04 and a frequency for System Damping of 2.377 Hz is applied in the analysis. The frequency was extracted from a normal mode analysis. The effects of gravity are not considered in this model.

Another run is performed with Rayleigh damping, using a Rayleigh damping stiffness matrix scale factor, alpha, of  $1.339 \text{ E}-3$ , and a Rayleigh damping mass matrix scale factor, beta, of  $2.99 \text{ E}-1$ .

### Dynamic Parameters

Number of Time Steps: 2000

Time per Step: 0.0002

Output Intervals: 0

### Solution Type

Transient Dynamic/Time History

### Solution Method

Direct Transient

### Comparison of Results

The tabular results for the run with the structural damping are given in Table 1, while the tabular results for the run with the Rayleigh damping are given in Table 2.

**Table 1. Results for Structural Damping**

| Description                      | Theory | Autodesk Nastran | Error (%) |
|----------------------------------|--------|------------------|-----------|
| Peak Displacement (mm)           | 3.523  | 3.459            | 1.8       |
| Time of Peak Displacement (s)    | 0.210  | 0.210            | 0.0       |
| Peak Stress (N/mm <sup>2</sup> ) | 2.484  | 2.431            | 0.3       |

**Table 2. Results for Rayleigh Damping**

| Description                      | Theory | Autodesk Nastran | Error (%) |
|----------------------------------|--------|------------------|-----------|
| Peak Displacement (mm)           | 3.523  | 3.459            | 1.8       |
| Time of Peak Displacement (s)    | 0.210  | 0.210            | 0.0       |
| Peak Stress (N/mm <sup>2</sup> ) | 2.484  | 2.432            | 0.3       |

## References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

## 8.7 Direct Frequency Response – Simply Supported Plate

### Problem Description

Figure 1 shows the plate model. Two direct frequency response analyses are performed on the model, with structural damping and another one with Rayleigh damping. Peak deflection and stress are determined, as well as frequencies. All dimensions are in meters.

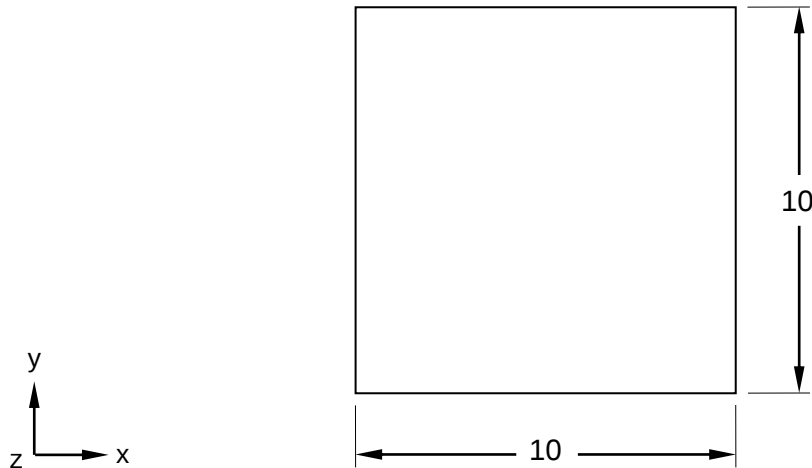


Figure 1. Plate Model

### Autodesk Nastran Analysis Model Filename

- vm8\_7.nas

### Model Data

#### *Finite Element Modeling*

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 10.0$  m

Width:  $w = 10.0$  m

Thickness:  $t = 0.05$  m

#### *Cross Sectional Properties*

Area:  $A = 100.0$  m<sup>2</sup>

Square Cross Section = (10.0 m x 10.0 m)

### Material Properties

Young's Modulus:  $E = 200.0 \text{ E}+9 \text{ N/m}^2$

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 8000 \text{ kg/m}^3$

### Boundary Conditions

The boundary conditions are as follows: fixed  $T_x$ ,  $T_y$ , and  $R_z$  at all nodes; fixed  $T_z$  along all 4 edges; fixed  $R_x$  along edges  $X = 0$  and  $X = 10 \text{ m}$  (left and right edges); fixed  $R_y$  along  $Y = 0$  and  $Y = 10 \text{ m}$  (top and bottom edges). A frequency forcing function with pressure load of magnitude  $100 \text{ N/m}^2$  is applied on the entire plate. An overall structural damping coefficient ( $G$ ) of 0.04 is applied. The effects of gravity are not considered in this model.

Another run is performed with Rayleigh damping, using a Rayleigh damping stiffness matrix scale factor, alpha, of  $1.339\text{E}-3$ , and a Rayleigh damping mass matrix scale factor, beta, of  $2.99 \text{ E}-1$ .

### Solution Type

Frequency/Harmonic Response

### Solution Method

Direct Frequency

### Comparison of Results

The tabular results for the run with the structural damping are given in Table 1, while the tabular results for the run with the Rayleigh damping are given in Table 2.

**Table 1. Results for Structural Damping**

| Description                      | Theory | Autodesk Nastran | Error (%) |
|----------------------------------|--------|------------------|-----------|
| Peak Displacement (mm)           | 45.42  | 45.24            | 0.4       |
| Frequency (Hz)                   | 2.377  | 2.377            | 0.0       |
| Peak Stress (N/mm <sup>2</sup> ) | 30.03  | 31.63            | 5.1       |

**Table 2. Results for Rayleigh Damping**

| Description                      | Theory | Autodesk Nastran | Error (%) |
|----------------------------------|--------|------------------|-----------|
| Peak Displacement (mm)           | 45.42  | 45.22            | 0.4       |
| Frequency (Hz)                   | 2.377  | 2.377            | 0.0       |
| Peak Stress (N/mm <sup>2</sup> ) | 30.03  | 31.61            | 5.0       |

### References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

## 9. Dynamics Verification Using Theoretical Solutions

The purpose of these dynamic test cases is to verify the functionality of Autodesk Nastran using theoretical solutions of well-known engineering dynamic problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

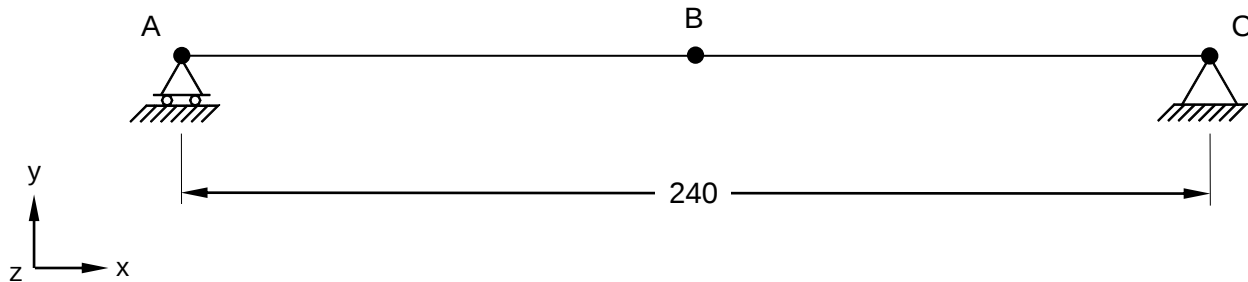
The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

## 9.1 Seismic Response of a Beam Structure

### Problem Description

Figure 1 shows the beam model. Response spectrum analysis is performed on the model. Assume zero damping and lumped mass. The fundamental frequency, fundamental displacement, and the maximum bending stress for the beam structure subjected to a seismic displacement response spectrum are determined. All dimensions are in inches.



**Figure 1. Beam Model**

### Autodesk Nastran Analysis Model Filename

- vm9\_1.nas

### Model Data

#### *Finite Element Modeling*

- 26 nodes, 25 beam elements

#### *Units*

inch/pond/second

#### *Model Geometry*

Length:  $L = 240$  in

Height:  $h = 14$  in

#### *Cross Sectional Properties*

Area:  $A = 273.9726$  in<sup>2</sup>

#### *Material Properties*

Young's Modulus:  $E = 30.0$  E+6 psi

Poisson's Ratio:  $\nu = 0.33$

Mass:  $m = 0.2$  lb-sec<sup>2</sup>/in

### Boundary Conditions

One end of the beam (point A) is restrained in Z-translation, X and Y-rotations, while the other end of the beam (point C) is restrained in X and Z-translations, and X and Y-rotations. All other nodes are restrained in Z-translation, X and Y-rotations. The middle node of the beam (point B) is also constrained in the Y-translation. A response spectrum dynamic loading is applied as follows:

| Frequency (Hz) | Displacement (in) |
|----------------|-------------------|
| 0.1            | 0.44              |
| 10.0           | 0.44              |

### Solution Type

Modal Summation

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                  | Theory | Autodesk Nastran | Error (%) |
|------------------------------|--------|------------------|-----------|
| Frequency (Hz)               | 6.0979 | 6.0979           | 0.0       |
| Displacement (in)            | -0.560 | -0.5594          | 0.1       |
| Maximum Bending Stress (psi) | 20,158 | 20,158           | 0.0       |

### References

1. Biggs, J. M., *Introduction to Structural Dynamics*. New York: McGraw-Hill Co., 1964.



## 9.2 Cantilever Beam Subjected to Sine Plus Tip Load

### Problem Description

Figure 1 shows the beam model. A transient dynamic time history analysis using time history forcing functions is performed on the model. The peak displacement at the tip of the beam due to a half-sine pulse tip load is determined. All dimensions are in inches.

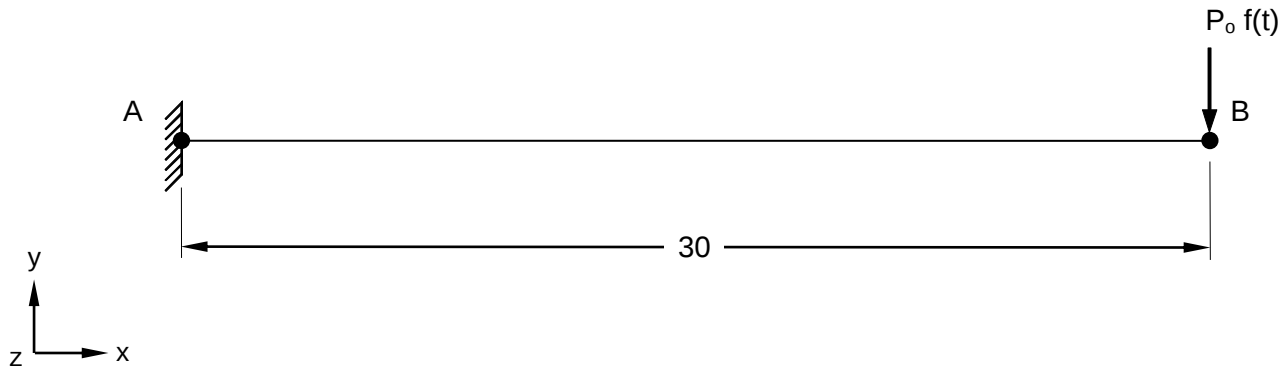


Figure 1. Beam Model

### Autodesk Nastran Analysis Model Filename

- vm9\_2.nas

### Model Data

#### *Finite Element Modeling*

- 7 nodes, 6 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 30$  in

#### *Cross Sectional Properties*

Area:  $A = 0.04$  in<sup>2</sup>

Moment of Inertia:  $I = 1.33333$  E-4 in<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 10.0$  E+6 psi

Poisson's Ratio:  $\nu = 0.3$

Mass Density:  $\rho = 0.1$  lbf/in<sup>3</sup>

### Boundary Conditions

The beam is loaded with a forcing function which is half-sine pulse:

$$P_o f(t)$$

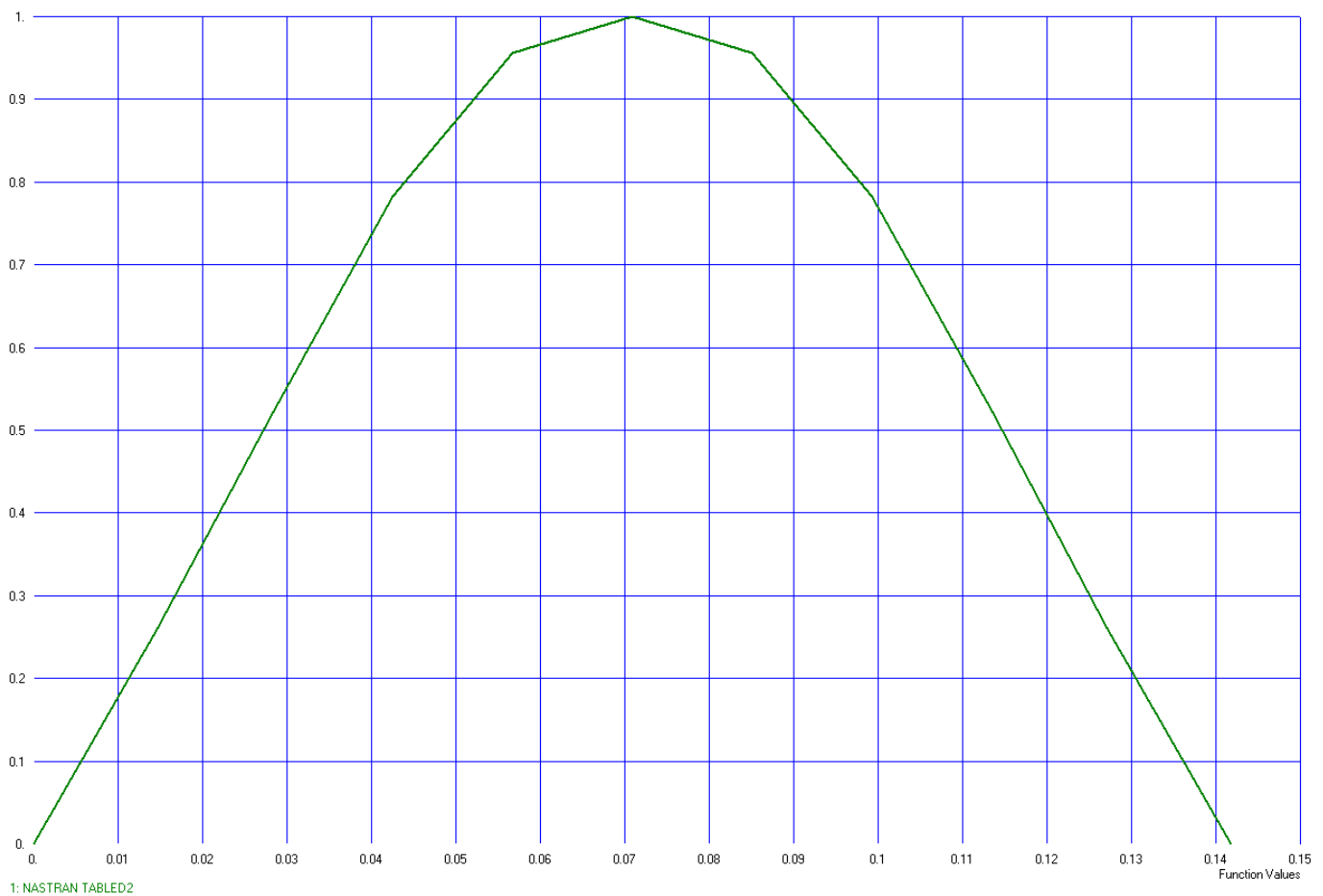
where:

$$P_o = -0.1b$$

$$f(t) = \sin \pi t / T$$

$$T = 2\pi / \omega_1$$

The input function is shown in Figure 2.



**Figure 2. Input Loading Function**

One end of the beam (point A) is restrained in all translations and rotations. All other nodes are restrained in Z-translation, X and Y-rotations.

**Dynamic Parameters**

Number of Time Steps: 283

Time per Step: 0.001

Output Intervals: 1

**Solution Type**

Modal Transient Response

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description       | Theory | Autodesk<br>Nastran | Error (%) |
|-------------------|--------|---------------------|-----------|
| Displacement (in) | -1.151 | -1.148              | 0.3       |

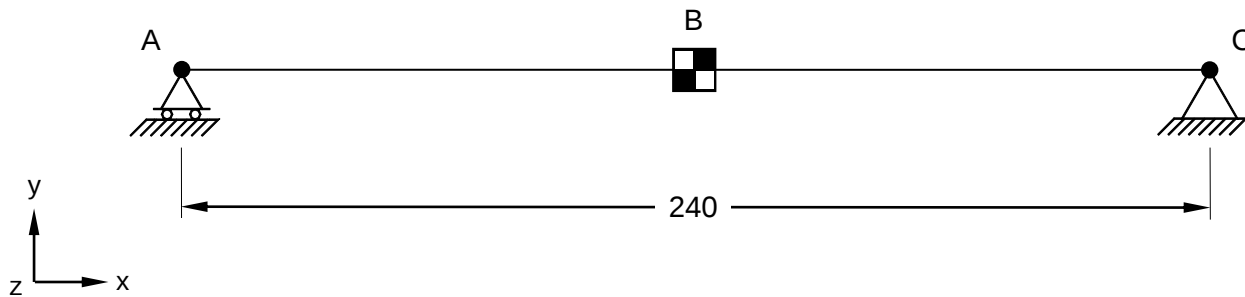
**References**

1. Warburton, G. B., *The Dynamic Behavior of Structures*. Pergamon Press, 1964.

### 9.3 Simply Supported Beam, Ramped Nodal Forcing Function, Transient Forced Vibration

#### Problem Description

Figure 1 shows the beam model. A transient dynamic time history analysis using time history forcing functions is performed on the model. The peak displacement at the tip of the beam and the time at which the pick occurs, and the peak dynamic stress are determined. All dimensions are in inches.



**Figure 1. Beam Model**

#### Autodesk Nastran Analysis Model Filename

- vm9\_3.nas

#### Model Data

##### *Finite Element Modeling*

- 3 nodes, 3 beam elements and 1 mass element

##### *Units*

inch/pound/second

##### *Model Geometry*

Length:  $L = 240$  in

Width:  $w = 18$  in

##### *Cross Sectional Properties*

Area:  $A = 180$  in<sup>2</sup>

Moment of Inertia:  $I = 800.6$  in<sup>4</sup>

##### *Material Properties*

Young's Modulus:  $E = 30.0$  E+6 psi

Poisson's Ratio:  $\nu = 0.3$

Mass:  $m = 25.9067$  lbf-sec<sup>2</sup>/in

**Boundary Conditions**

One end of the beam (point A) is restrained in the Y and Z-translations, and X and Y-rotations, while the other end of the beam (point C) is restrained in all translations, and X and Y-rotations. The middle node of the beam (point B) is also constrained in the Z-translation, X and Y-rotations. The beam weight is assumed negligible. A mass  $m = 25.9067 \text{ lbf-sec}^2/\text{in}$  is considered at the center of the beam. A time step force data is applied as follows:

| Time (sec) | Force (lbf) |
|------------|-------------|
| 0.000      | 0           |
| 0.075      | 20000       |
| 0.100      | 20000       |

**Dynamic Parameters**

Number of Time Steps: 100

Time per Step: 0.001

Output Intervals: 1

**Solution Type**

Modal Transient Response

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                   | Theory | Autodesk Nastran | Error (%) |
|-------------------------------|--------|------------------|-----------|
| Peak Displacement (in)        | 0.331  | 0.336            | 1.5       |
| Time of Peak Displacement (s) | 0.092  | 0.093            | 1.1       |
| Peak Stress (psi)             | 18,600 | 18,438           | 0.9       |

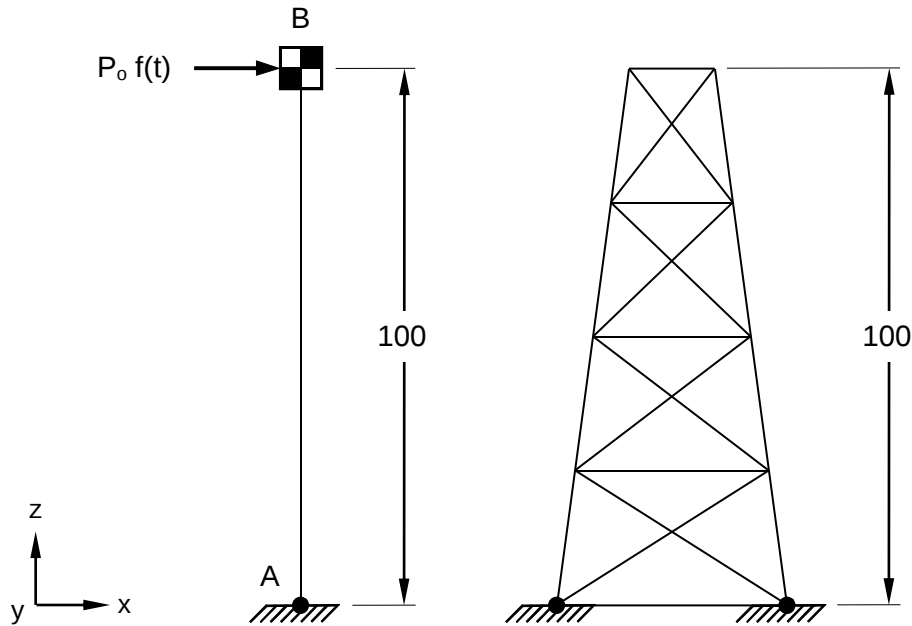
**References**

1. Biggs, J. M., *Introduction to Structural Dynamics*. New York: McGraw-Hill Co., 1964.

## 9.4 Tower Structure Under a Harmonic Excitation Force

### Problem Description

Figure 1 shows the tower structure and its beam equivalent. A transient dynamic time history analysis using time history forcing functions is performed on the structure that is subjected to a lateral harmonic excitation force. The peak displacement at the tip due to a half-sine pulse tip load and the first natural mode are determined. All dimensions are in inches.



**Figure 1. Tower Model**

### Autodesk Nastran Analysis Model Filename

- vm9\_4.nas

### Model Data

#### *Finite Element Modeling*

- 11 nodes, 10 beam elements and 1 mass element

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 100$  in

#### *Cross Sectional Properties*

Area:  $A = 1$  in<sup>2</sup>

Moment of Inertia:  $I = 1666.667$  in<sup>4</sup>

### Material Properties

Young's Modulus:  $E = 20.0 \text{ E}+6 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

Mass:  $m = 100 \text{ lbf-sec}^2/\text{in}$  (in the X-direction)

### Boundary Conditions

A harmonic excitation force is applied at the top node (point B) in the X-direction:

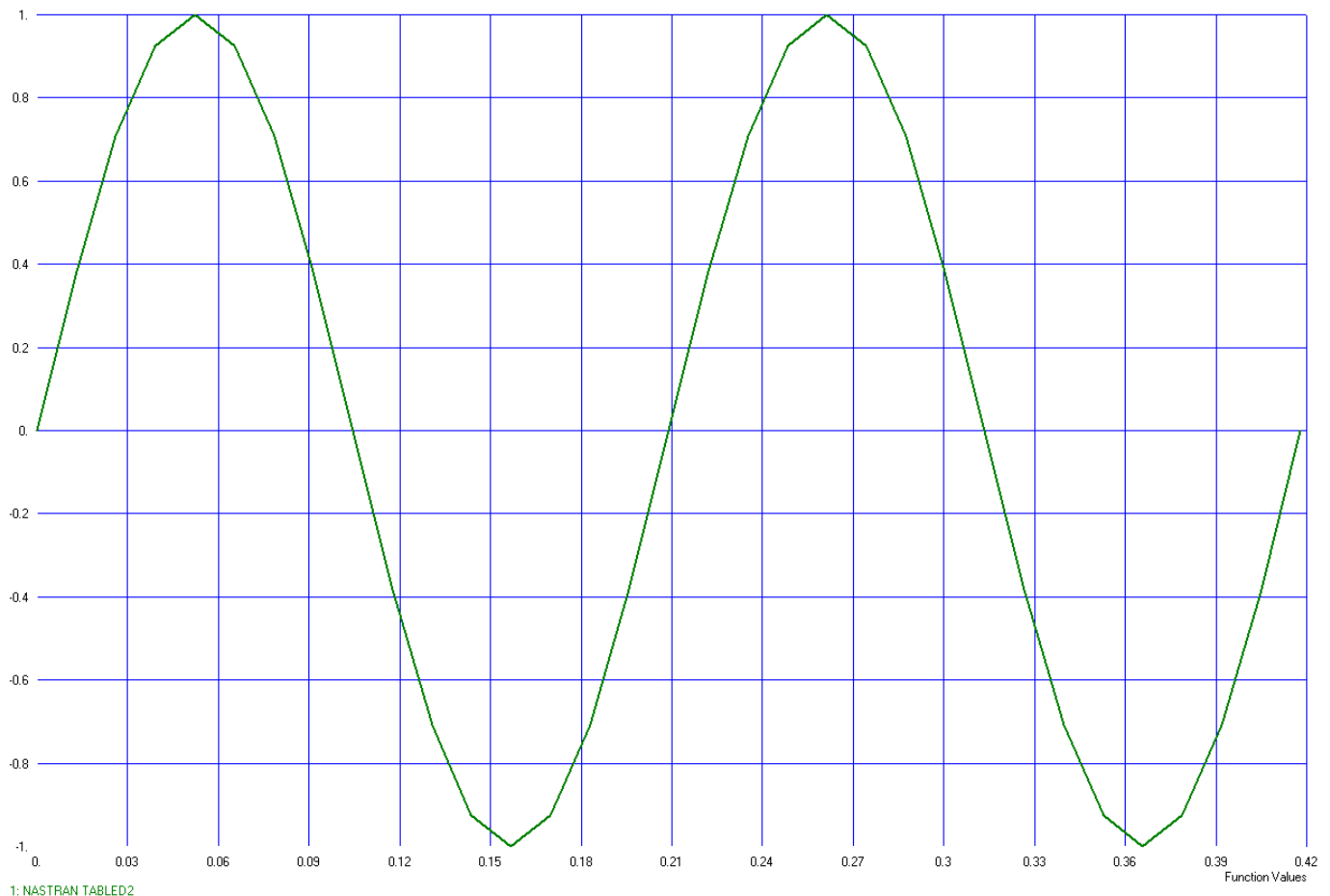
$$P(t) = P_o \sin \omega t \text{ lbf}$$

where:

$$P_o = 1.0 \text{ E} + 5 \text{ lbf}$$

$$\omega = 30 \text{ rad/sec}$$

The input function is shown in Figure 2.



**Figure 2. Input Loading Function**

One end of the beam (point A) is restrained in all translations and rotations, while all the other nodes are restrained in Y and Z-translations, and all rotations. A mass  $m = 100 \text{ lbf-sec}^2/\text{in}$  is considered at the top of the beam. Damping ratio is zero.

**Dynamic Parameters**

Number of Time Steps: 300

Time per Step: 0.001

Output Intervals: 1

**Solution Type**

Modal Transient Response

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                               | Theory | Autodesk Nastran | Error (%) |
|-------------------------------------------|--------|------------------|-----------|
| Peak Displacement (in) at t = 0.1 sec     | 1.608  | 1.585            | 0.1       |
| Peak Displacement (in) at t = 0.2 sec     | -3.187 | -3.146           | 1.3       |
| Peak Displacement (in) at t = 0.3 sec     | 4.742  | 4.655            | 1.8       |
| First Natural Frequency (cycles/sec=1/Hz) | 0.200  | 0.199            | 0.5       |

**References**

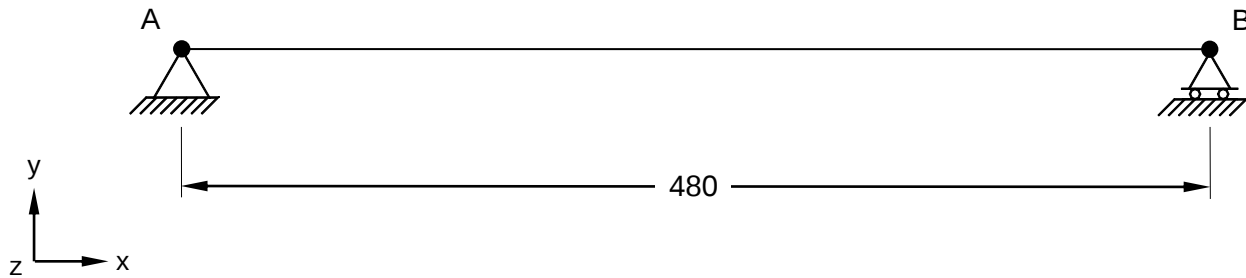
1. Paz, M., *Structural Dynamics; Theory and Computation*. 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1991.



## 9.5 Simply Supported Beam Subjected to a Traveling Dynamic Load

### Problem Description

Figure 1 shows the simply supported beam model. A transient dynamic time history analysis using time history forcing functions and multiple arrival times is performed on the model that is subjected to a dynamic force traveling along the span at a constant velocity. The displacements at the mid-span of the beam with a time history are determined. All dimensions are in inches.



**Figure 1. Beam Model**

### Autodesk Nastran Analysis Model Filename

- vm9\_5.nas

### Model Data

#### *Finite Element Modeling*

- 21 nodes, 20 beam elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 480$  in

#### *Cross Sectional Properties*

Area:  $A = 1$  in<sup>2</sup>

Moment of Inertia:  $I = 0.083333$  in<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 2.4 \text{ E}+11$  psi

Poisson's Ratio:  $\nu = 0.3$

Specific Weight:  $\gamma = 0.1$  lbf/in<sup>3</sup>

**Boundary Conditions**

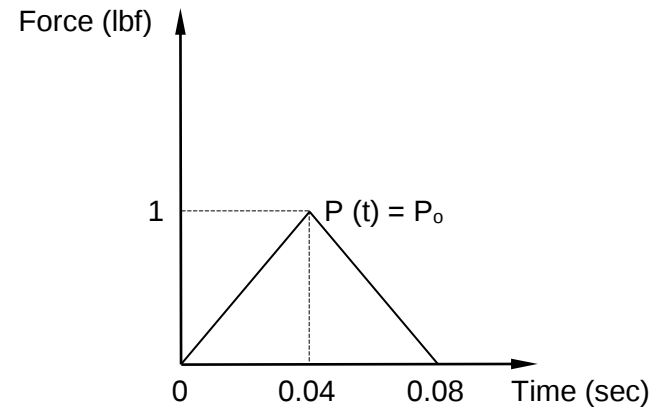
A triangular excitation force is applied in the Z-direction and travels at 600 in/sec in the X-direction:

$P(t) = P_o \text{ lbf}$

where:

$P_o = 1.0 \text{ lbf}$

The triangular load is shown in Figure 2.



**Figure 2. Input Loading Function**

One end of the beam (point A) is restrained in all translations, X and Y- rotations, while the other end of the beam (point B) is restrained in the Y and Z-translations, X and Y-rotations. All the other nodes are restrained in the Z-translations, and X and Y-rotations. Damping ratio is zero.

**Dynamic Parameters**

Number of time steps: 800

Time per step: 0.001

Output Intervals: 1

**Solution Type**

Modal Transient Response

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                     | Theory | Autodesk Nastran | Error (%) |
|---------------------------------|--------|------------------|-----------|
| Displacement at mid span (in)   | -1.100 | 1.087            | 1.2       |
| Time at Max. Displacement (sec) | 0.280  | 0.311            | 11.1      |
| Velocity at mid span (in/sec)   | 7.500  | 7.896            | 5.3       |
| Time at Max. Velocity (sec)     | 0.670  | 0.712            | 6.3       |

## References

1. Biggs, J. M., *Introduction to Structural Dynamics*. New York: McGraw-Hill Co., 1964.

## 10. Nonlinear Static Verification Using Theoretical Solutions

The purpose of these nonlinear static test cases is to verify the functionality of Autodesk Nastran using theoretical solutions of nonlinear static problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are either from reputable engineering texts or are calculated by Autodesk, Inc. For each applicable case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

## 10.1 Nonlinear Cable Tension

### Problem Description

Figure 1 shows the cable model. A nonlinear static analysis is performed on the model. The cable force of the tensioned cable is determined. All dimensions are in inches.

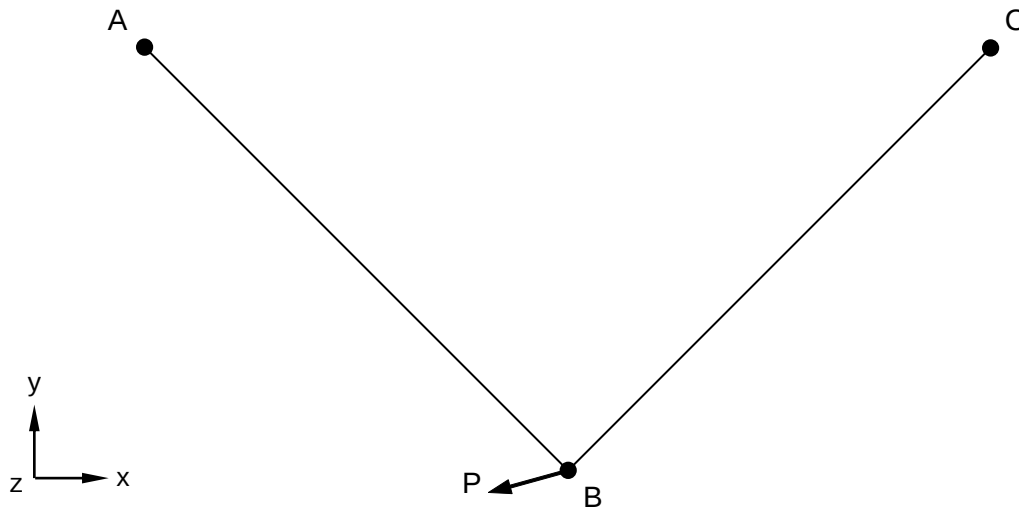


Figure 1. Cable Model

### Autodesk Nastran Analysis Model Filename

- vm10\_1.nas

### Model Data

#### *Finite Element Modeling*

- 3 nodes, 2 1-DOF/node cable elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:

Global X component = 2347.5 in

Global Y component = 2000.0 in

#### *Cross Sectional Properties*

Area:  $A = 706 \text{ in}^2$

### Boundary Conditions

The cables are loaded at the joining node with a point load  $P = 8417.2$  lb. The load's X-component  $F_x$  is 7700 lb in the negative direction, while the Y-component  $F_y$  is 3400 lb in the negative direction. The upper ends of the cables (points A and C) are constrained in all translations and rotations except the Z-rotation. The joined end of the cables (point B) is constrained in the Z-translation and X and Y-rotations.

### Nonlinear Parameters

Static

Number of Increments: 5

### Solution Type

Nonlinear Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                         | Theory  | Autodesk Nastran | Error (%) |
|-------------------------------------|---------|------------------|-----------|
| Cable Force of Tensioned Cable (lb) | 8,417.2 | 8,417.4          | 0.0       |

**Note:** The theoretical solution is simply the input load. One cable should slacken while the other takes all of the load, thus the cable force should also equal the input force.

## 10.2 Cable Supporting Hanging Loads

### Problem Description

Figure 1 shows the cable model. Nonlinear static analysis is performed on the plate. The horizontal and vertical reaction forces at support A and the maximum tension in the cable are determined. All dimensions are in feet.

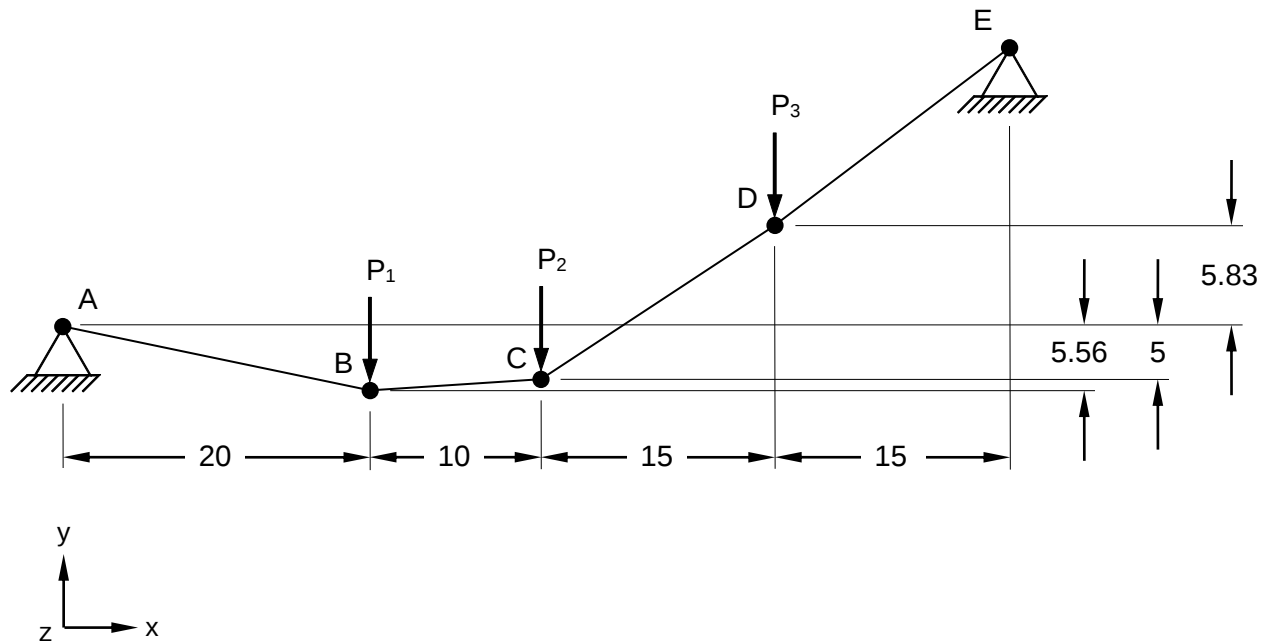


Figure 1. Cable Model Supporting Hanging Loads

### Autodesk Nastran Analysis Model Filename

- vm10\_2.nas

### Model Data

#### Finite Element Modeling

- 5 nodes, 4 1-DOF/node cable elements

#### Units

feet/Kip/second

#### Model Geometry

Length:  $L = 60$  ft (total horizontal length)

#### Cross Sectional Properties

Area:  $A = 0.1 \text{ ft}^2$

### Material Properties

Young's Modulus:  $E = 2.0 \text{ E}+7 \text{ ksf}$

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

All translations and rotations are restrained for point A. Points B, C and D have the Z-translation, and the X and Y-rotations constrained, while point E is restrained in all translations. Vertical point loads  $P1 = 6 \text{ Kip}$ ,  $P2 = 12 \text{ Kip}$ , and  $P3 = 4 \text{ Kip}$  are applied to the cable model (see Figure 1).

### Nonlinear Parameters

Static

Number of Increments: 1

### Solution Type

Nonlinear Static

### Comparison of Results

Tabular results are given in Table 1.

**Table 1. Results**

| Description                          | Theory | Autodesk Nastran | Error (%) |
|--------------------------------------|--------|------------------|-----------|
| Reaction Force in X-direction (Kips) | 18.00  | 18.00            | 0.0       |
| Reaction Force in Y-direction (Kips) | 5.00   | 5.00             | 0.0       |
| Cable Force (Kips)                   | 24.76  | 24.76            | 0.0       |

### References

1. Beer, F. P., and Johnston, Jr. E. R., *Vector Mechanics for Engineers, Statics and Dynamics*. New York: McGraw-Hill, Inc., 1962.



## 10.3 Ten Story Plane Frame

### Problem Description

Figure 1 shows the plane frame model. Nonlinear static analysis is performed on the beam (to simulate the P-delta analysis with 2 iterations). The maximum lateral displacement of the 10<sup>th</sup> story after 2 iterations of the P-delta analysis is determined. All dimensions are in inches.

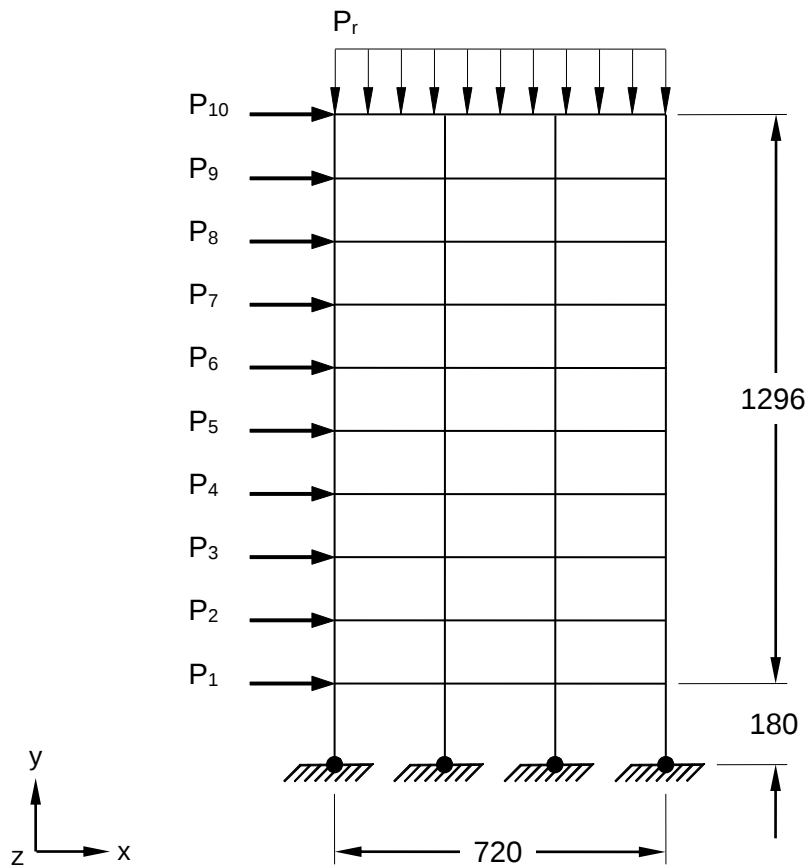


Figure 1. Ten Story Plane Frame Model

### Autodesk Nastran Analysis Model Filename

- vm10\_3.nas

### Model Data

#### *Finite Element Modeling*

- 44 nodes, 70 beam elements

#### *Units*

inch/Kip/second

**Model Geometry**

Length:  $L = 720$  in (3 x 240 in)

Height:  $h = 1476$  in (1 x 180 in and 9 x 144 in)

**Material Properties**

Young's Modulus:  $E = 2.9 \text{ E}+7$  psi

Poisson's Ratio:  $\nu = 0.3$

**Boundary Conditions**

All translations and rotations are restrained at supports. All the other nodes are restrained in the Z-translation and X and Y-rotations. A typical roof gravity load  $P_r = 0.25$  Kip/in and floor gravity load  $P_f = 0.30$  Kip/in (on each floor) are applied to the frame. Also point loads are applied to the left hand side beams, as shown in Figure 1:  $P_1 = 2.79$  Kip,  $P_2 = 5.34$  Kip,  $P_3 = 7.71$  Kip,  $P_4 = 10.08$  Kip,  $P_5 = 12.45$  Kip,  $P_6 = 14.83$  Kip,  $P_7 = 17.20$  Kip,  $P_8 = 19.57$  Kip,  $P_9 = 21.94$  Kip and  $P_{10} = 30.22$  Kip.

**Nonlinear Parameters**

Static

Number of Increments: 1

**Solution Type**

Nonlinear Static

**Comparison of Results**

Tabular results are given in Table 1.

**Table 1. Results**

| Description                                         | Theory | Autodesk Nastran | Error (%) |
|-----------------------------------------------------|--------|------------------|-----------|
| Lateral Displacement at 10 <sup>th</sup> Story (in) | 8.51   | 8.61             | 1.2       |

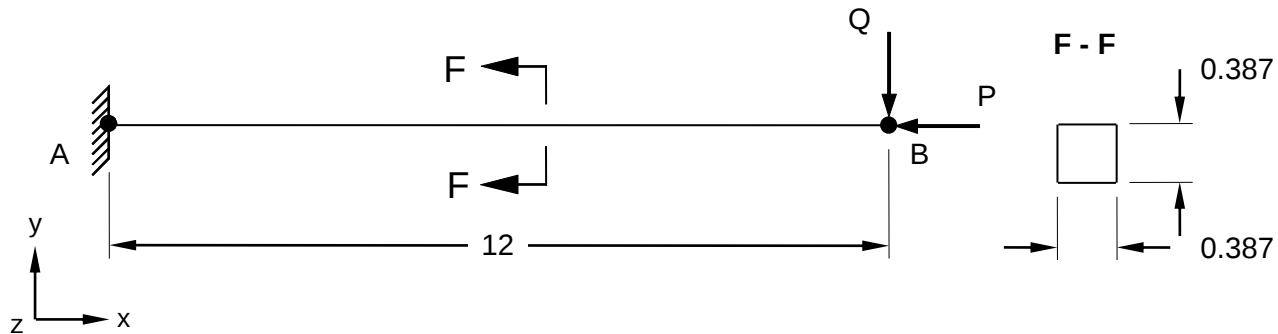
**References**

1. Naeim, F., *The Seismic Design Handbook*. New York: VanNostrand Reinhold Co., 1989.

## 10.4 Straight Cantilever with Axial End Point Load

### Problem Description

Figure 1 shows the cantilever model. A nonlinear static analysis is performed on the model. The free end tip displacement and rotations at different load increments are determined. All dimensions are in inches.



**Figure 1. Cantilever Model**

### Autodesk Nastran Analysis Model Filename

- vm10\_4.nas

### Model Data

#### *Finite Element Modeling*

- 13 nodes, 12 bar elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 12$  in

#### *Cross Sectional Properties*

Area:  $A = 0.15$  in<sup>2</sup>

Square Cross Section = (0.387 in x 0.387 in)

#### *Material Properties*

Young's Modulus:  $E = 2.0 \text{ E}+7$  psi

Shear Modulus of Elasticity:  $G = 1.0 \text{ E}+7$  psi

Poisson's Ratio:  $\nu = 0.0$

### Boundary Conditions

The cantilever is constrained in all translations and rotations at point A. All other nodes are constrained in the Z-translation, X and Y-rotations. An axial force (P) and shear force (Q) at the free end are applied in increments up to a maximum value of  $P/P_{cr} = 9.116$ , where  $P_{cr} = (\pi^2 EI) / 4L^2 = 96.4$  pounds, and  $Q = P/10$ .

### Nonlinear Parameters

Static

Number of Increments: 15

### Solution Type

Nonlinear Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                                    | Theory | Autodesk Nastran | Error (%) |
|----------------------------------------------------------------|--------|------------------|-----------|
| Applied Force $P/P_{cr} = 1.152$                               | 1.152  | 1.152            | N/A       |
| Applied Force $P/P_{cr} = 2.541$                               | 2.541  | 2.541            | N/A       |
| Applied Force $P/P_{cr} = 9.116$                               | 9.116  | 9.116            | N/A       |
| Tip Displacement in X-direction $U_x/L$ ( $P/P_{cr} = 1.152$ ) | 0.741  | 0.743            | 0.3       |
| Tip Displacement in X-direction $U_x/L$ ( $P/P_{cr} = 2.541$ ) | 0.107  | 0.105            | 1.9       |
| Tip Displacement in X-direction $U_x/L$ ( $P/P_{cr} = 9.116$ ) | 0.575  | 0.575            | 0.0       |
| Tip Displacement in Y-direction $U_y/L$ ( $P/P_{cr} = 1.152$ ) | 0.593  | 0.592            | 0.2       |
| Tip Displacement in Y-direction $U_y/L$ ( $P/P_{cr} = 2.541$ ) | 0.750  | 0.752            | 0.3       |
| Tip Displacement in Y-direction $U_y/L$ ( $P/P_{cr} = 9.116$ ) | 0.421  | 0.424            | 0.7       |
| Tip Rotation $\phi$ ( $P/P_{cr} = 1.152$ ) (degrees)           | 60.0   | 59.8             | 0.3       |
| Tip Rotation $\phi$ ( $P/P_{cr} = 2.541$ ) (degrees)           | 140.0  | 140.0            | 0.0       |
| Tip Rotation $\phi$ ( $P/P_{cr} = 9.116$ ) (degrees)           | 176.0  | 176.1            | 0.0       |

### References

1. Timoshenko, S. and Gere, J. M., *Theory of Elastic Stability*, 3<sup>rd</sup> Edition. New York: McGraw-Hill Book Co., Inc., 1970.

## 10.5 Residual Stress Problem

### Problem Description

Figure 1 shows the three tie rods model. A nonlinear static analysis is performed on the model. The displacement at point A under load P1 and the residual stress in the center rod at load P2, where the three rods become fully plastic and then the load is fully released, are determined. All dimensions are in inches.

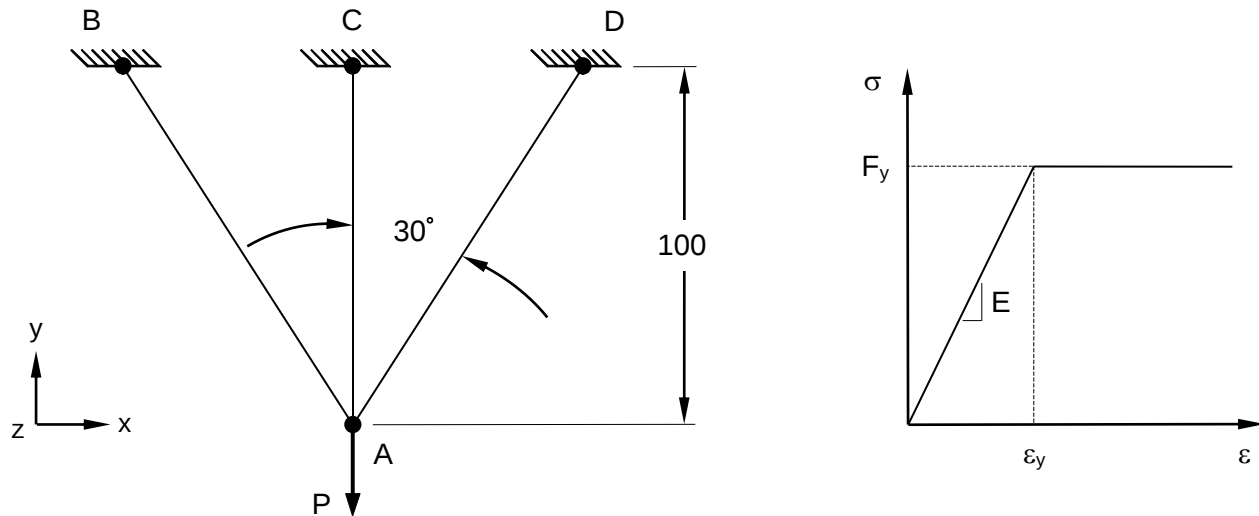


Figure 1. Three Tie Rods Model

### Autodesk Nastran Analysis Model Filename

- vm10\_5.nas

### Model Data

#### *Finite Element Modeling*

- 4 nodes, 3 rod elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 100$  in

#### *Cross Sectional Properties*

Area:  $A = 1$  in<sup>2</sup>

### Material Properties

Young's Modulus:  $E = 30.0 \text{ E}+6 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

Yield Force:  $F_y = 30.0 \text{ E}+3 \text{ psi}$

### Boundary Conditions

All the top nodes (points B, C, and D) are constrained in all translations and rotations. A force  $P_1 = 51,961.52422 \text{ lb}$  and  $P_2 = 81,961.52422 \text{ lb}$  is applied at point A in the negative Y-direction; then unload the applied load in the same increments.

### Nonlinear Parameters

Static

Number of Increments: 50

### Solution Type

Nonlinear Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                        | Theory    | Autodesk Nastran | Error (%) |
|------------------------------------|-----------|------------------|-----------|
| Tip Deflection @ Point A (in)      | -7.533E-2 | -7.531E-2        | 0.0       |
| Residual Stress @ Center Rod (psi) | -5,660    | -5,639           | 0.4       |

### References

1. Crandall, S. H., and Dahl, N. C., *An Introduction to the Mechanics of Solids*. New York: McGraw-Hill Book Co., Inc., 1959.

## 11. Nonlinear Static Verification Using Standard NAFEMS Benchmarks

The purpose of these nonlinear static test cases is to verify the functionality of Autodesk Nastran using benchmarks published by NAFEMS (National Agency for Finite Element Methods and Standards, National Engineering Laboratory, Glasgow, U.K.).

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

## 11.1 Elastic Large Deformation Response of a Z-shaped Cantilever Under an End Load

### Problem Description

Figure 1 shows the Z-shaped cantilever model. A nonlinear static analysis is performed on the model. The tip displacement at different load increments is determined. All dimensions are in inches.

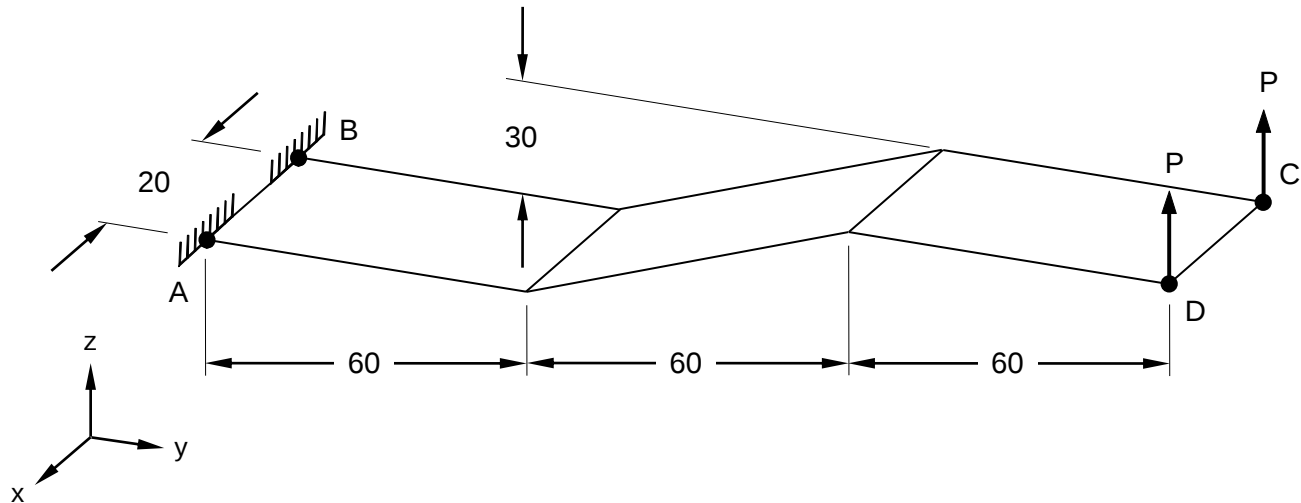


Figure 1. Z-shaped Cantilever Model

### Autodesk Nastran Analysis Model Filename

- vm11\_1.nas

### Model Data

#### *Finite Element Modeling*

- 95 nodes, 72 5-DOF/node quadrilateral plate elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length:  $L = 2000$  in

Width:  $w = 20$  in

Thickness:  $t = 1.7$  in

#### *Material Properties*

Young's Modulus:  $E = 2.05 \text{ E}+5$  psi

Poisson's Ratio:  $\nu = 0.25$



**Boundary Conditions**

The plate is constrained in all translations and rotations along edge AB. A uniform load  $P = 4000$  lbs is applied at the free end (edge CD) in the positive Z-direction.

**Nonlinear Parameters**

Static

Number of Increments: 100

**Solution Type**

Nonlinear Static

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                     | Theory | Autodesk Nastran | Error (%) |
|-------------------------------------------------|--------|------------------|-----------|
| Tip Displacement at 104.5 lbs Applied Load (in) | 80.42  | 78.43            | 2.5       |
| Tip Displacement at 1263 lbs Applied Load (in)  | 133.10 | 132.10           | 0.8       |
| Tip Displacement at 4000 lbs Applied Load (in)  | 143.50 | 142.70           | 0.6       |

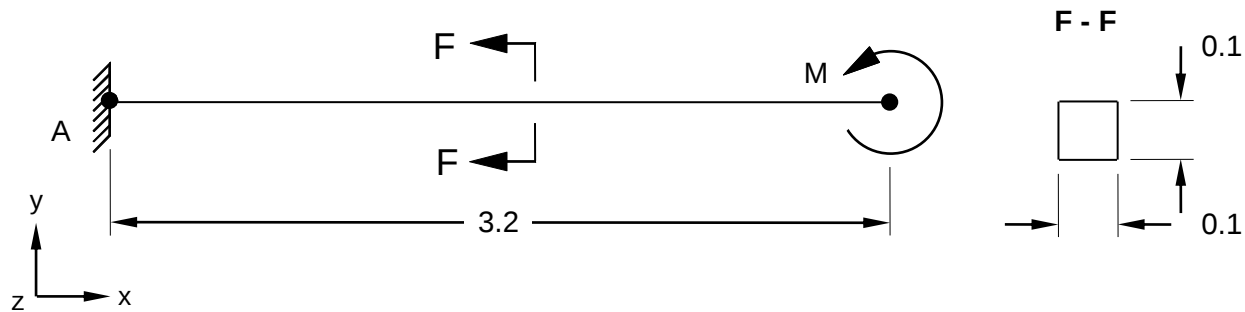
**References**

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Geometric Non-Linear Behavior of Shells*. Glasgow: NAFEMS, Publication R0024. Test 3DNLG-1.

## 11.2 Straight Cantilever with End Moment

### Problem Description

Figure 1 shows the cantilever beam model. A nonlinear static analysis is performed on the model. The free end tip displacement and rotations at different load increments are determined. All dimensions are in meters.



**Figure 1. Cantilever Model**

### Autodesk Nastran Analysis Model Filename

- vm11\_2.nas

### Model Data

#### *Finite Element Modeling*

- 21 nodes, 20 beam elements

#### *Units*

meter/Newton/second

#### *Model Geometry*

Length:  $L = 3.2$  m

Width:  $w = 0.1$  m

Thickness:  $t = 0.1$  m

#### *Cross Sectional Properties*

Area:  $A = 0.01$  m<sup>2</sup>

Moment of Inertia:  $I = 8.333 \text{ E-}6$  m<sup>4</sup>

#### *Material Properties*

Young's Modulus:  $E = 210.0$  GPa

Shear Modulus of Elasticity:  $G = 105.0$  GPa

Poisson's Ratio:  $\nu = 0.0$

### Boundary Conditions

The beam is constrained in all translations and rotations at point A. All other nodes are constrained in Z-translation, X and Y-rotations. A moment  $M$  is applied at the free end in increments up to  $ML/2\pi EI = 1.0$ . Thus the moment  $M = 1.0 \times 2 \times \pi \times E \times I / L = 1.0 \times 2 \times \pi \times 210 \times 10^9 \times 8.333 \times 10^{-6} / 3.2 = 3435979.0$

### Nonlinear Parameters

Static

Number of Increments: 50

### Solution Type

Nonlinear Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                                                  | Theory | Autodesk Nastran | Error (%) |
|--------------------------------------------------------------|--------|------------------|-----------|
| Applied Moment $ML/2\pi EI$ (at increment = 0.5)             | 0.5    | 0.5              | N/A       |
| Applied Moment $ML/2\pi EI$ (at increment = 1.0)             | 1.0    | 1.0              | N/A       |
| Tip Displacement in X-direction $U_x/L$ (at increment = 0.5) | 1.0    | 1.0              | 0.0       |
| Tip Displacement in X-direction $U_x/L$ (at increment = 1.0) | 1.0    | 1.0              | 0.0       |
| Tip Displacement in Y-direction $U_y/L$ (at increment = 0.5) | 0.637  | 0.637            | 0.0       |
| Tip Displacement in Y-direction $U_y/L$ (at increment = 1.0) | 0.0    | 0.0              | 0.0       |
| Tip Rotation $\phi/2\pi$ (at increment = 0.5)                | 0.5    | 0.5              | 0.0       |
| Tip Rotation $\phi/2\pi$ (at increment = 1.0)                | 1.0    | 1.0              | 0.0       |

### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Non-Linear Behavior of 3D-Beams*. Glasgow: NAFEMS, Publication NNB, Rev. 1, Oct. 1989. Test NL5.

## 11.3 Lee's Frame Buckling Problem

### Problem Description

Figure 1 shows the right angle frame model. A nonlinear static analysis is performed on the model. The displacement at point of loading at different load increments are determined. All dimensions are in meters.

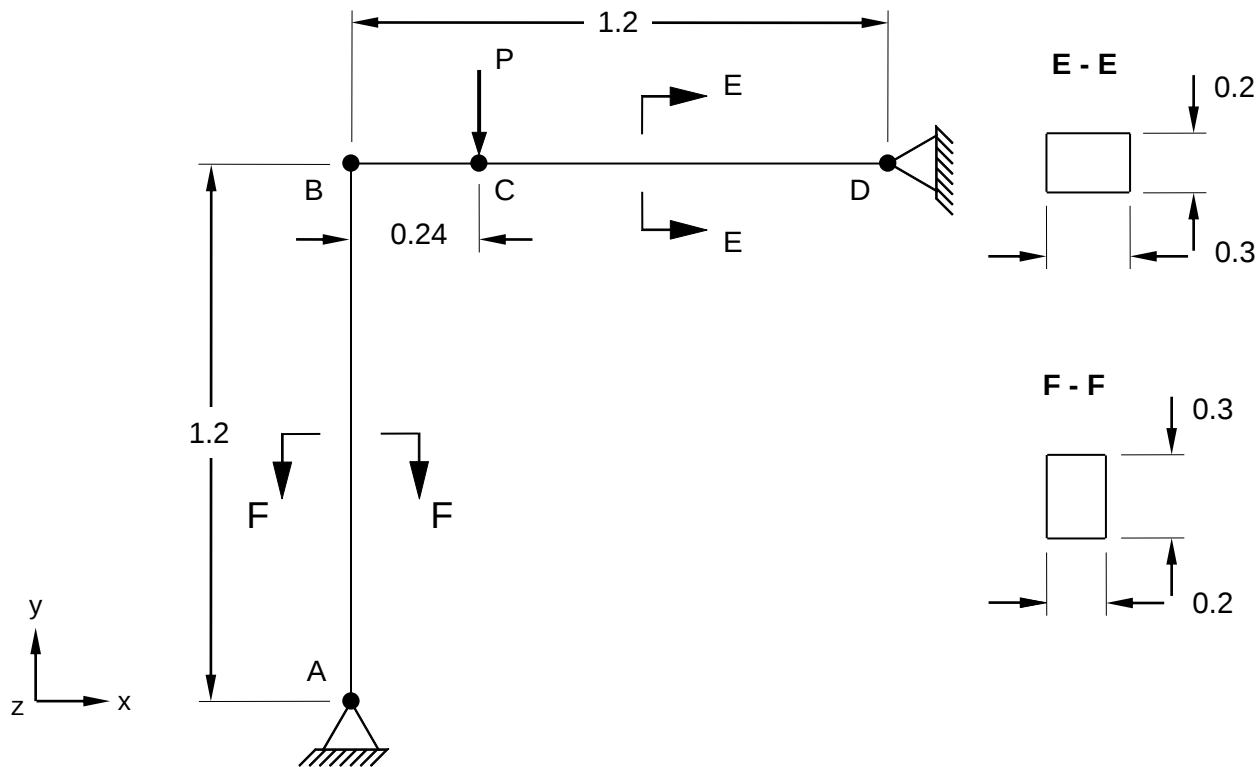


Figure 1. Right Angle Frame Model

### Autodesk Nastran Analysis Model Filename

- vm11\_3.nas

### Model Data

#### Finite Element Modeling

- 21 nodes, 20 beam elements

#### Units

meter/Newton/second

#### Model Geometry

Length:  $L = 1.2$  m

### Cross Sectional Properties

Square Cross Section 1 = (0.03 m x 0.02 m)

Square Cross Section 2 = (0.02 m x 0.03 m)

### Material Properties

Young's Modulus:  $E = 71.74$  GPa

Poisson's Ratio:  $\nu = 0.3$

### Boundary Conditions

Points A and D are constrained in all translations, X and Y-rotations. All other nodes are constrained in the Z-translation, X and Y-rotations. A shear force P is applied at point C in increments up to a maximum value of  $PL^2/EI = 31.887$ .

### Nonlinear Parameters

Static

Number of Increments: 50

### Solution Type

Nonlinear Static

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                             | Theory | Autodesk Nastran | Error (%) |
|-----------------------------------------|--------|------------------|-----------|
| Applied Force $PL^2/EI$                 | 18.552 | 18.552           | N/A       |
| Applied Force $PL^2/EI$                 | 31.887 | 31.887           | N/A       |
| Tip Displacement in Y-direction $U_y/L$ | 0.407  | 0.408            | 0.2       |
| Tip Displacement in Y-direction $U_y/L$ | 0.784  | 0.782            | 0.3       |

### References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Non-Linear Behavior of 3D-Beams*. Glasgow: NAFEMS, Publication NNB, Rev. 1, Oct. 1989. Test NL5.

## 12. Nonlinear Dynamic Verification Using Theoretical Solutions

The purpose of these nonlinear dynamic test cases is to verify the functionality of Autodesk Nastran using theoretical solutions of well-known engineering nonlinear dynamic problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

## 12.1 Impact Load on a Rod by a Mass at a Constant Velocity

### Problem Description

Figure 1 shows the rod and mass model. A nonlinear transient analysis with impact loading is performed on the model. The strain energy, time to contact and maximum stress on the bar are determined. All dimensions are in inches.

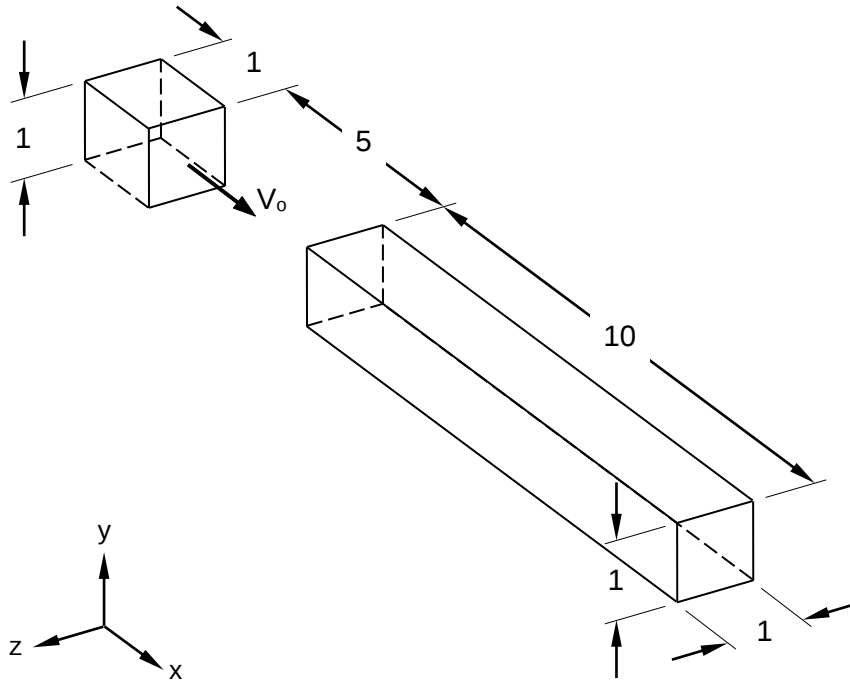


Figure 1. Rod and Mass Model

### Autodesk Nastran Analysis Model Filename

- vm12\_1.nas

### Model Data

#### *Finite Element Modeling*

- 240 nodes, 117 hex solid elements, and 16 gap elements

#### *Units*

inch/pound/second

#### *Model Geometry*

Length Rod:  $L_r = 10$  in

Length Mass:  $L_m = 1$  in

Distance Mass to Rod:  $L = 5$  in

Width:  $w = 1$  in

Height:  $h = 1$  in

**Cross Sectional Properties**

Area Rod:  $A_r = 1 \text{ in}^2$

Area Mass:  $A_m = 1 \text{ in}^2$

Square Cross Section Rod = (1 in x 1 in)

Square Cross Section Mass = (1 in x 1 in)

**Material Properties**

Young's Modulus Rod:  $E_r = 30.0 \text{ E}+6 \text{ psi}$

Young's Modulus Mass:  $E_m = 1.0 \text{ E}+5 \text{ psi}$

Poisson's Ratio:  $\nu = 0.3$

**Physical Properties**

Mass Rod:  $m_r = 0.0001 \text{ lb-sec/in}$

Mass Moving Body:  $m_b = 0.2588 \text{ lb-sec/in}$

**Boundary Conditions**

One end of the beam is constrained in all translations and all rotations. The mass has the Y and Z-translations, and all rotations constrained, to be guided in the X-direction to impact on the beam. An initial velocity  $V_0 = 100 \text{ in/sec}$  is applied to the mass.

**Nonlinear Parameters**

Transient

Subcase 1:

Number of Increments: 50

Time Increment: 0.001

Subcase 2:

Number of Increments: 200

Time Increment: 0.0001

**Solution Type**

Nonlinear Transient (impact loading)

**Comparison of Results**

The tabular results are given in Table 1.

**Table 1. Results**

| Description                 | Theory | Autodesk Nastran | Error (%) |
|-----------------------------|--------|------------------|-----------|
| Strain Energy $U_m$ (in-lb) | 1,294  | 1,366            | 5.6       |
| Time to Contact (sec)       | 5E-2   | 5E-2             | 0.0       |
| Maximum Stress (psi)        | 44,057 | 44,551           | 1.1       |



## References

1. Beer, F. P. and Johnston, Jr. E. R., *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1981.

## 12.2 Impact of a Block on a Spring Scale

### Problem Description

Figure 1 shows the rod and mass model. A nonlinear transient analysis with impact loading is performed on the model. The maximum deflection of the scale pan, time to contact and total deflection of the block are determined. All dimensions are in inches.

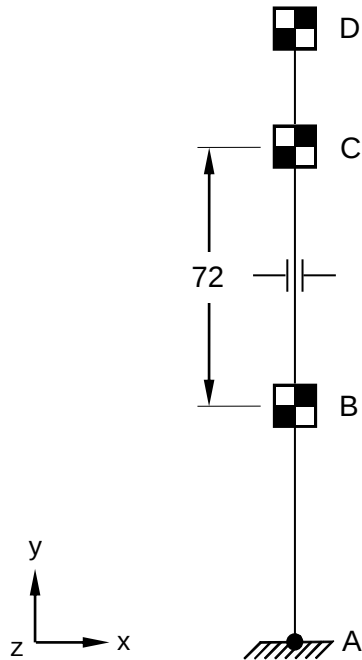


Figure 1. Rod and Mass Model

### Autodesk Nastran Analysis Model Filename

- vm12\_2.nas

### Model Data

#### *Finite Element Modeling*

- 4 nodes, 1 rod element (AB), 3 mass elements, 1 rigid body element (CD), 1 gap element (BC)

#### *Units*

inch/pound/second

#### *Model Geometry*

Distance Block to Scale:  $h = 72$  in

### Material Properties

Young's Modulus Rod:  $E_r = 10.0 \text{ E}+2 \text{ psi}$

Poisson's Ratio:  $\nu = 0.25$

Weight Rod (scale pan):  $W_p = 25.0 \text{ lbs}$

Weight Falling Body:  $W_b = 50.0 \text{ lbs}$

Spring Stiffness (scale):  $k_1 = 100.0 \text{ lbs/in}$

### Boundary Conditions

One end of the rod (point A) is constrained in all translations and all rotations. All other nodes are constrained in the X and Z-translations and all rotations. Gravity load is applied in the negative Y-direction (1G acceleration).

### Nonlinear Parameters

Transient

Subcase 1:

Number of Increments: 100

Time Increment: 0.006135

Subcase 2:

Number of Increments: 1000

Time Increment: 0.0001

### Solution Type

Nonlinear Transient (impact loading)

### Comparison of Results

The tabular results are given in Table 1.

**Table 1. Results**

| Description                    | Theory  | Autodesk Nastran | Error (%) |
|--------------------------------|---------|------------------|-----------|
| Deflection of Scale Pan (in)   | -7.700  | -7.445           | 3.3       |
| Total Deflection of Block (in) | -79.450 | -79.442          | 0.0       |
| Time to Contact                | 0.611   | 0.613            | 0.3       |

### References

1. Beer, F. P. and Johnston, Jr. E. R., *Vector Mechanics for Engineers*. New York: McGraw-Hill, Inc., 1962.

**APPENDIX A – REFERENCES**

1. ASME Pressure Vessel and Piping Division, Committee, on Computer Technology, *Program Verification and Qualification Library*, 1972.
2. Bathe, K. J. and Wilson, E. L., *Large Eigenvalue Problems in Dynamic Analysis*. Journal of the Engineering Mechanics Division, ASCE, Vol. 98, No. EM6, Paper 9433, 1972.
3. Beer, F. P. and Johnston, Jr. E. R., *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1981.
4. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.
5. Beer, F. P. and Johnston, Jr. E. R., *Vector Mechanics for Engineers, Statics and Dynamics*. New York: McGraw-Hill, Inc., 1962.
6. Belvins, R., *Formulas For Natural Frequency and Mode Shape*, 1<sup>st</sup> Edition. New York: VanNostrand Reinhold Co., 1979.
7. Biggs, J. M., *Introduction to Structural Dynamics*. New York: McGraw-Hill Co., 1964.
8. Crandall, S. H., and Dahl, N. C., *An Introduction to the Mechanics of Solids*. New York: McGraw-Hill Book Co., Inc., 1959.
9. Davies, G. A. O., Fenner, R. T., and Lewis, R. W., *Background to Benchmarks*. Glasgow: NAFEMS, 1993.
10. Harris, C. M. and Crede, C. E., *Shock and Vibration Handbook*. New York: McGraw-Hill, Inc., 1976.
11. Harris, C. O., *Introduction to Stress Analysis*. New York: The Macmillan Co., 1959.
12. Hsu, T. H., *Stress and Strain Data Handbook*. Houston, London, Paris, Tokyo: Gulf Publishing Company, Book Division.
13. McCormac, J. C., *Structural Analysis*, 3<sup>rd</sup> Edition. New York: In Text Educational Publishers, 1975.
14. McNeal, R. H. and Harder, R. C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*, North Holland, 1985.
15. Naeim, F., *The Seismic Design Handbook*. New York: VanNostrand Reinhold Co., 1989.
16. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Forced Vibration*. Glasgow: NAFEMS, Mar., 1993. Test No. 5T.
17. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Geometric Non-Linear Behavior of Shells*. Glasgow: NAFEMS, Publication R0024. Test 3DNLG-1.
18. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov. 1987.
19. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Non-Linear Behavior of 3D-Beams*. Glasgow: NAFEMS, Publication NNB, Rev. 1, Oct. 1989. Test NL5.
20. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990.
21. NAFEMS Report R0031, *Laminated Strip*. Test No. R0031/1, Date issued 17/12/98/1.
22. NAFEMS Report R0031, *Wrapped Thick Cylinder*. Test No. R0031/2, Date issued 17/12/98/1.
23. NASA SP-8007 Buckling of Thin-Walled Circular Cylinders.
24. Paz, M., *Structural Dynamics; Theory and Computation*. 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1991.
25. Peterson, F. E., *EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual*, Engineering Analysis Corporation, Berkeley, California, 1981.
26. Roark, R. and Young, W., *Formulas for Stress and Strain*, 5<sup>th</sup> Edition. New York: McGraw-Hill Co., 1975.

27. Scordelis, A. C. and Lo, K. S., *Computer Analysis of Cylindrical Shells*, Journal of the American Concrete Institute, Vol. 61, May 1964.
28. Seely, F. B. and Smith, J. O., *Advanced Mechanics of Materials*, 2<sup>nd</sup> Edition. John Wiley and Sons, 1955.
29. Shigley, J. and Mitchel, L., *Mechanical Engineering Design*, 4<sup>th</sup> Edition. New York: McGraw-Hill Co., 1983.
30. Societe Francaise des Mecaniciens, *Guide de validation des progiciels de calcul de structures*. Paris: Afnor Technique, 1990.
31. Thompson, W. T., *Vibration Theory and Applications*, 2<sup>nd</sup> Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.
32. Timoshenko, S. and Gere, J. M., *Theory of Elastic Stability*, 2<sup>nd</sup> Edition. New York: McGraw-Hill Book Co., Inc., 1961.
33. Timoshenko, S. and Gere, J. M., *Theory of Elastic Stability*, 3<sup>rd</sup> Edition. New York: McGraw-Hill Book Co., Inc., 1970.
34. Timoshenko, S. and Young, D. H., *Vibration Problems in Engineering*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955.
35. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1955.
36. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3<sup>rd</sup> Edition. New York: D. VanNostrand Co., Inc., 1956.
37. Tse, F., Morse, I., and Hinkle, R., *Mechanical Vibrations*, 2<sup>nd</sup> Edition. Boston: Allyn and Bacon, Inc., 1978.
38. Tse, F., Morse, I., and Hinkle, R., *Mechanical Vibrations, Theory and Applications*. Boston: Allyn and Bacon, Inc., 1978.
39. Young, W. C., *Roark's Formulas for Stress and Strain*, 6<sup>th</sup> Edition. New York: McGraw-Hill Co., 1989.
40. Warburton, G. B., *The Dynamic Behavior of Structures*. Pergamon Press, 1964.